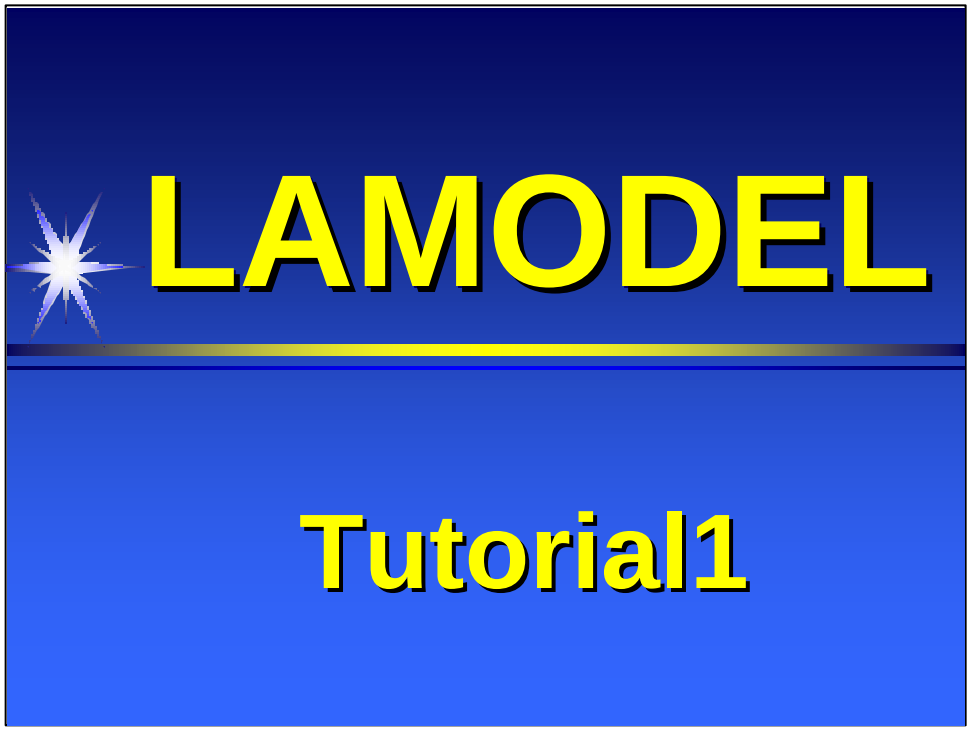
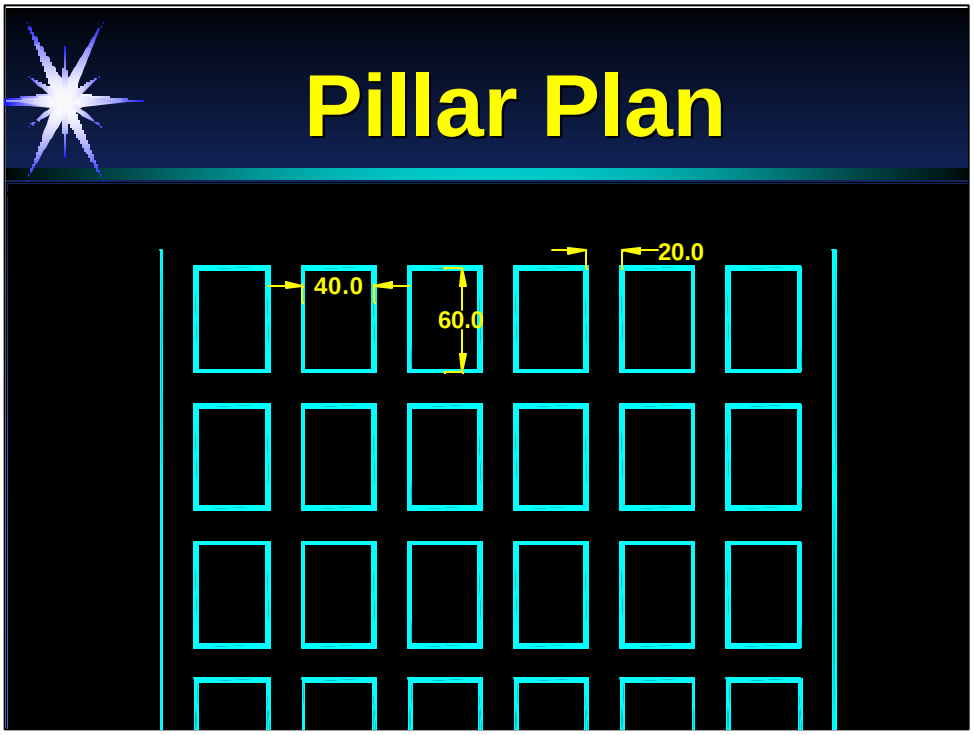


Let’s do a simple model using LAMODEL.

In this sample, we will do a very simple model and make use of many of the automatic and default features of the program.

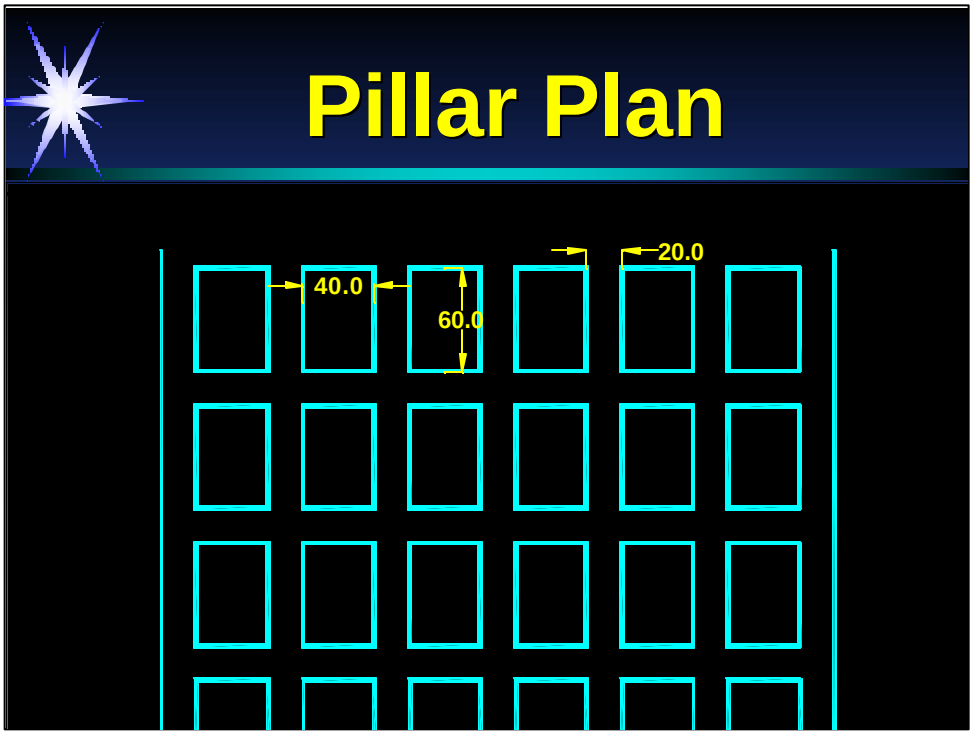
The very first step is to define the problem we want to investigate.



Let’s do a simple room-and-pillar production panel. First, we want to see the development loads, and then we want to look at the abutment stress’s when the pillars are pulled.

In this panel, we plan to have 7 entries each 20 ft wide with 40 by 60 ft pillars.

The coal seam is 5 ft thick and 700 ft deep.



For this mining plan, a 10 ft grid will work very well.

Also, if we make the total grid dimensions 100 by 100 elements, or 1000 by 1000 ft, we should be able to center the area of interest away from the grid boundary with room to spare.

To begin entering the parameters into the model, we need to start the preprocessor, LAMPRE.

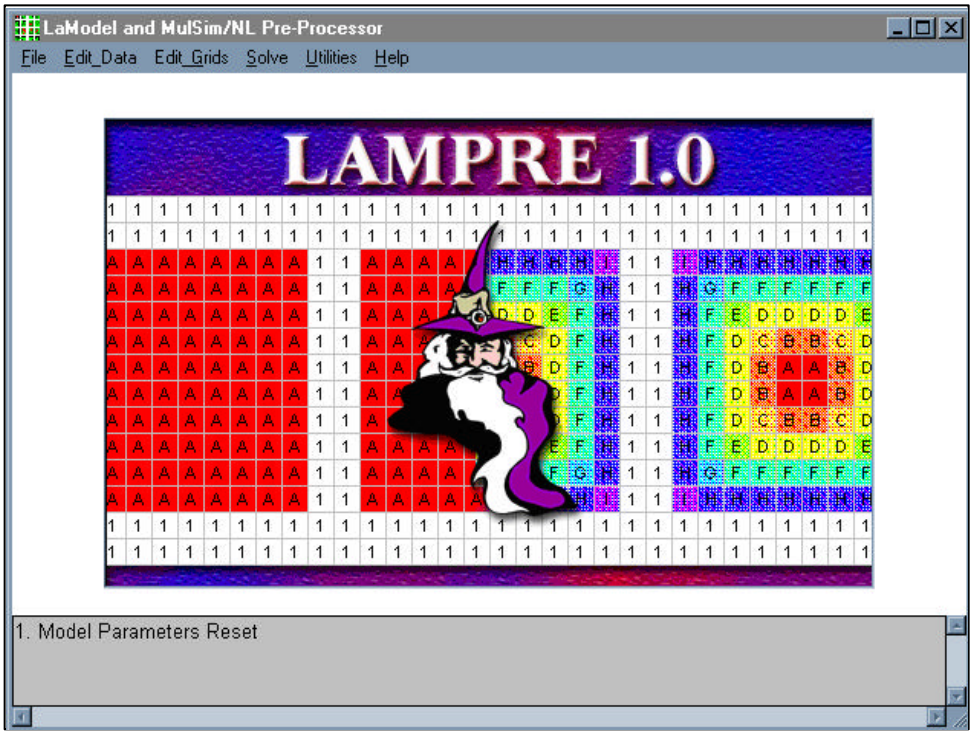
Click on:
Start=>Programs=>NIOSH=>LAMODEL=>LAMPRE

This will start LAMPRE, and the first thing that you will see is the Disclaimer.

Click on:

OK

And the main LAMPRE window will appear.

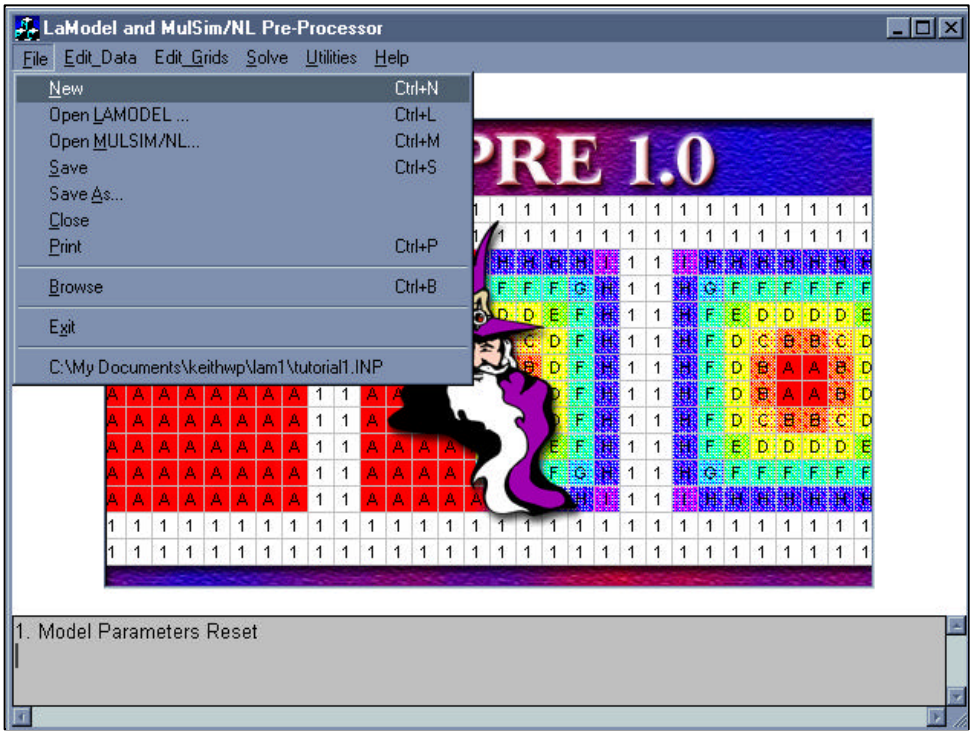


In the main window, you see a menu bar across the top, the LAMPRE 1.0 graphics in the middle and a history window at the bottom.

All of the parameter input is accomplished using the menu bar at the top of the window.

The history window at the bottom simple records the actions that the user has performed in this session.

Let’s take a closer look at the menu bar

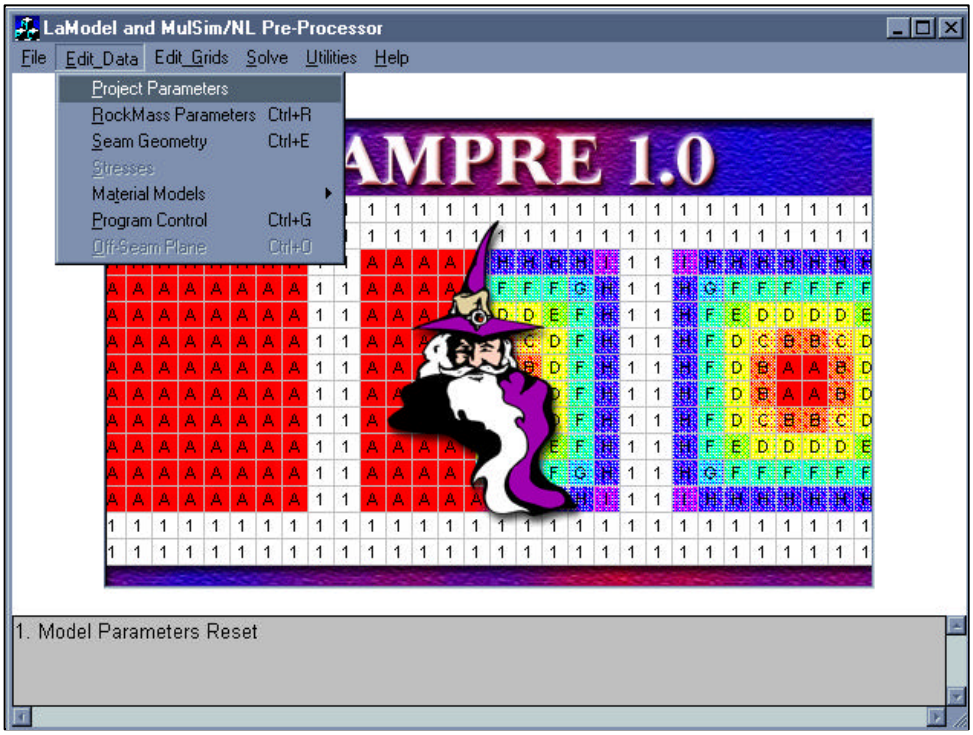


Click on: **File** on the menu bar

A fairly standard “Windows” file menu appears.

The user can: start a new file, open a LAMODEL input file, open a MULSIM input file, save, save as, close, exit, etc.

The window also remembers the last few files that were previously opened at the bottom.

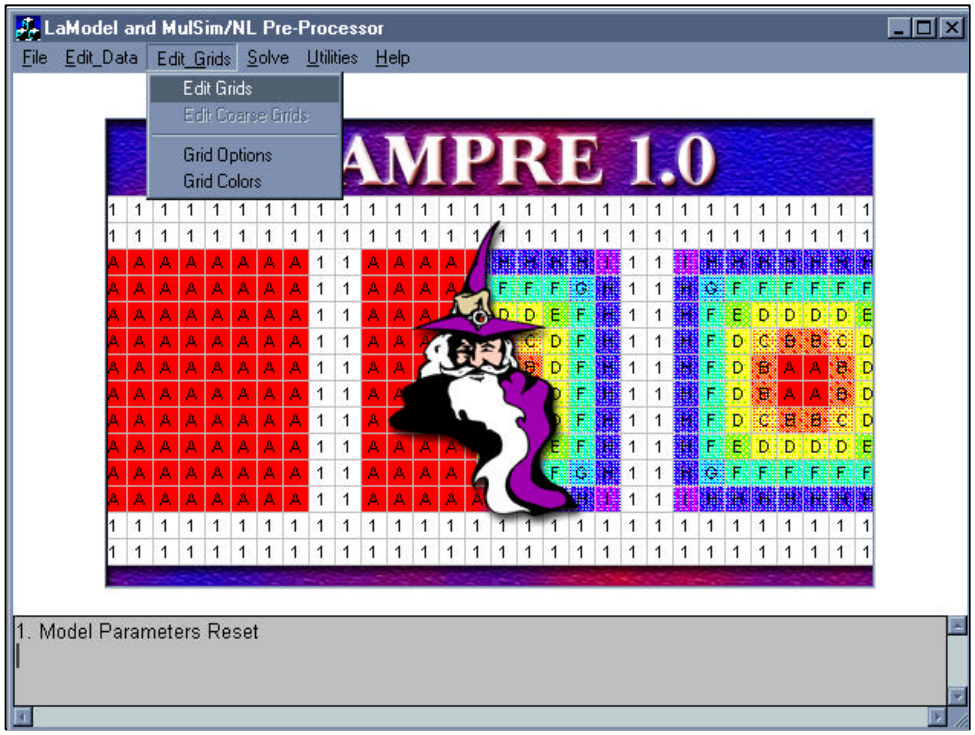


Click on: **Edit_Data** on the menu bar

The commands for the primary data input sub-menus appear.

These sub-menus are where the numerical parameters are entered into the input file for LAMODEL.

We will look at these parameter input menus in much more detail in a few minutes. First, let's move over to the Edit_Grids menu

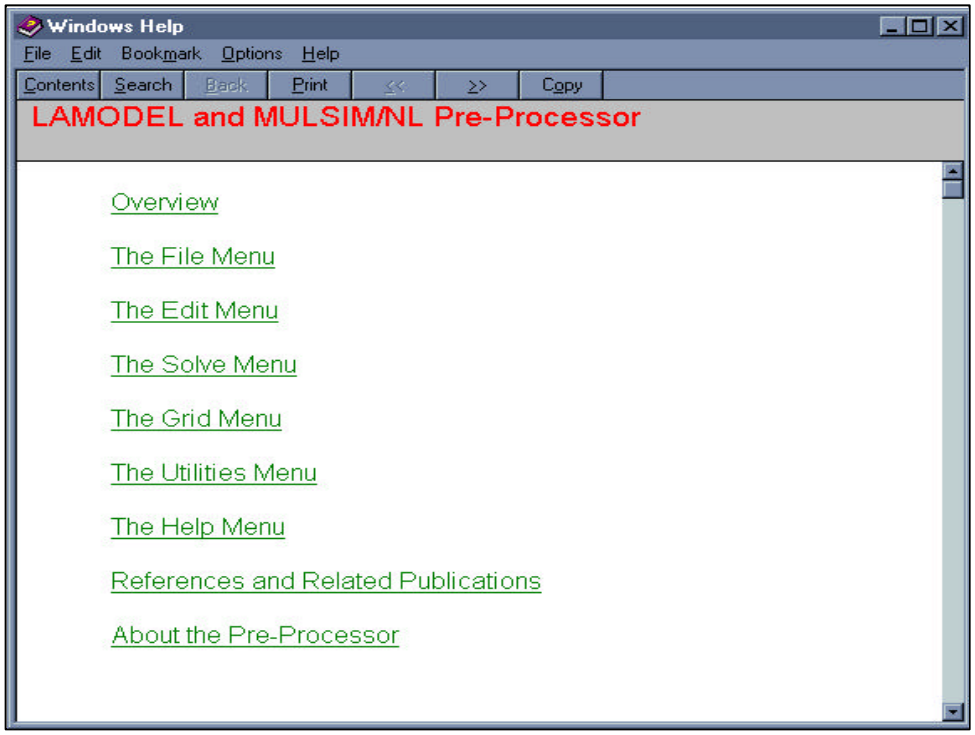


Click on: **Edit_Grids** on the menu bar

The sub-menus for editing the grids appear.

These sub-menus are where the element grids are graphically created for the input file to LAMODEL.

We will look at this graphical input menus in much more detail in a few minutes. Before that, let's take a quick look at the available help.

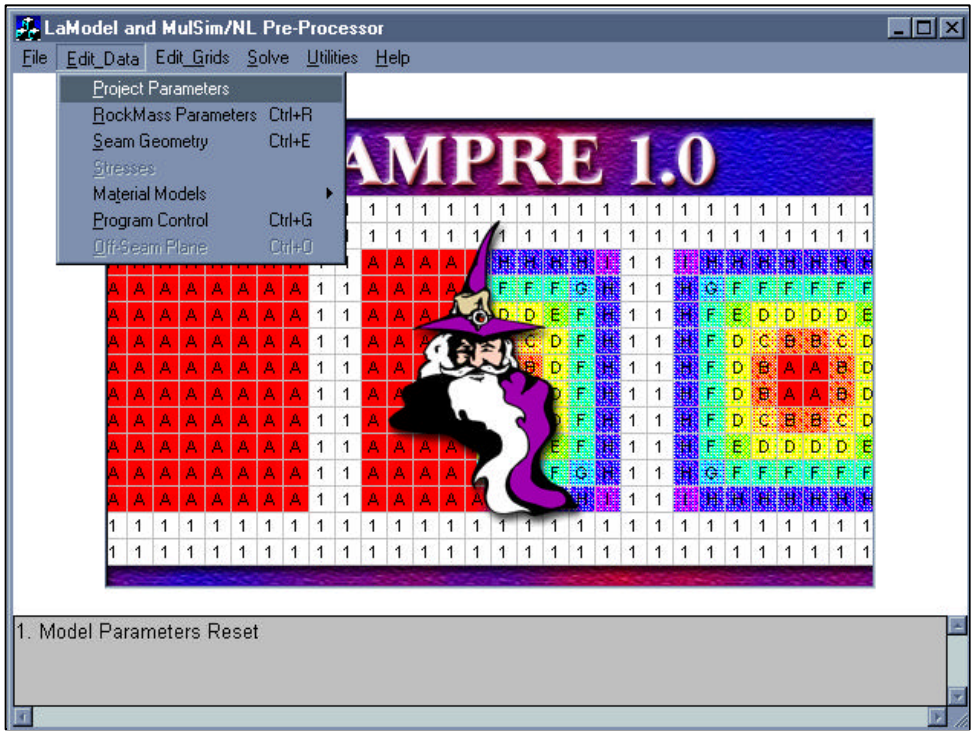


Click on: **Help=>Contents** on the menu bar

The contents of the help file appears.

LAMPRE contains fairly extensive help capabilities.

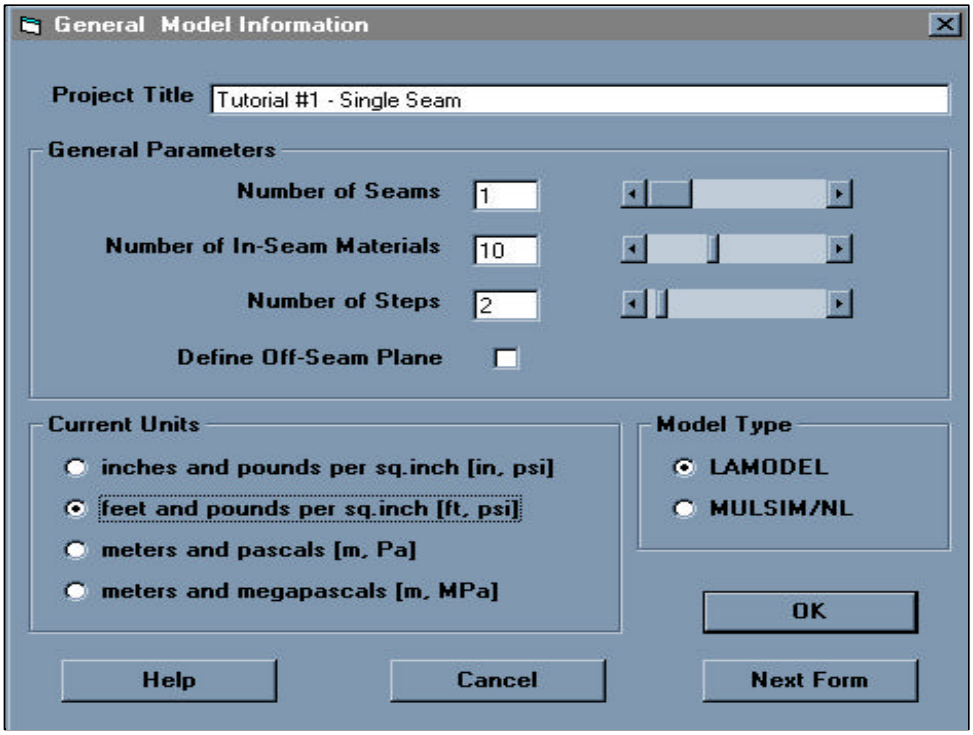
Every input parameter has an explanation and recommended values. The help for the individual parameters can be reached by working through the help hierarchy or by selecting that parameter in the sub-menu and pressing **"F1"**.



Let’s go back and start entering the model data.

We start by clicking on: **Edit_Data=>Project Parameters** from the menu bar

This brings up the “General Model Information” form.



In this form, we can enter a title, in this case **"Tutorial #1 – Single Seam".**

Next we set:

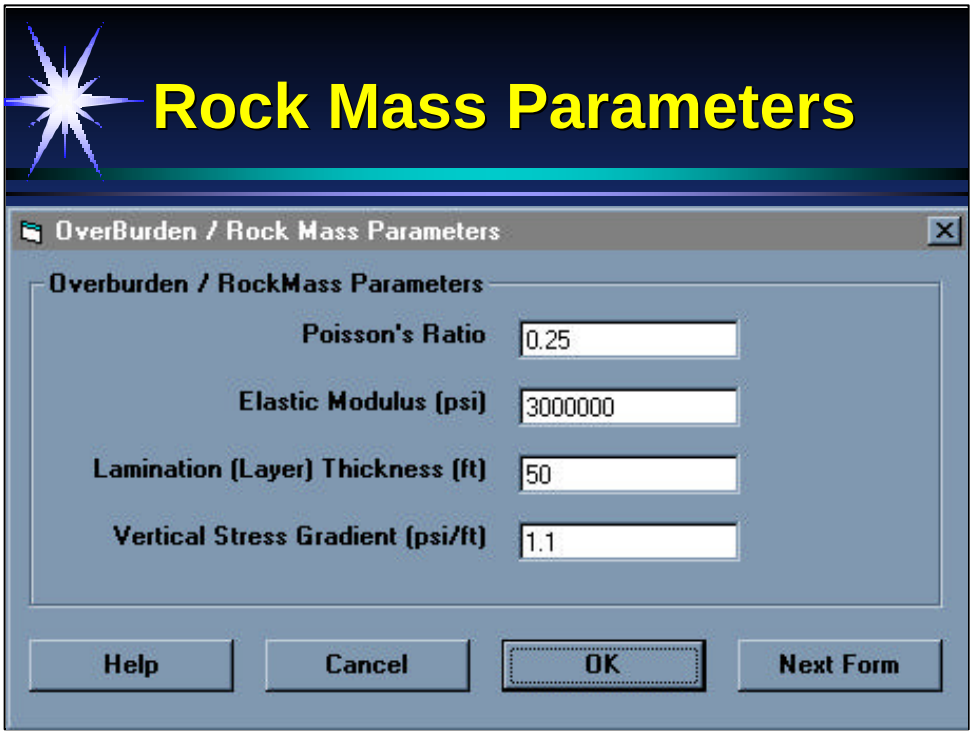
"1" Seam

"10" Materials (we will see later that this will allow us to use an automatic yield zone around the pillars)

"2" Steps – one for development and one for retreat mining

"ft, psi" – my preferred units

"LAMODEL" – for the model type.

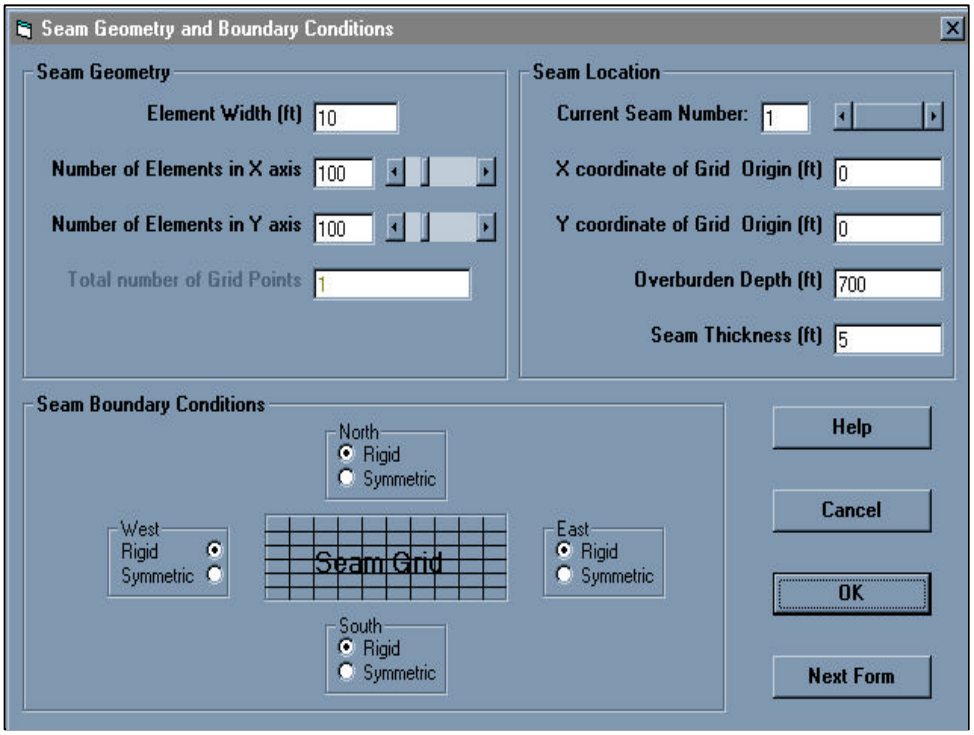


If we then click on: **Next Form,** this opens the **"Overburden / Rock Mass Parameters"** form.

If we had site-specific rock mass data, or some experience/engineering judgment, we might adjust these parameters to better fit our location.

However, since we do not have any site-specific information, we can just use the default values and we will get "reasonable" overburden response.

To continue, we click on: **Next Form**



This opens the **"Seam Geometry and Boundary Conditions"** form.

As discussed earlier, we will use:

"10" ft element widths

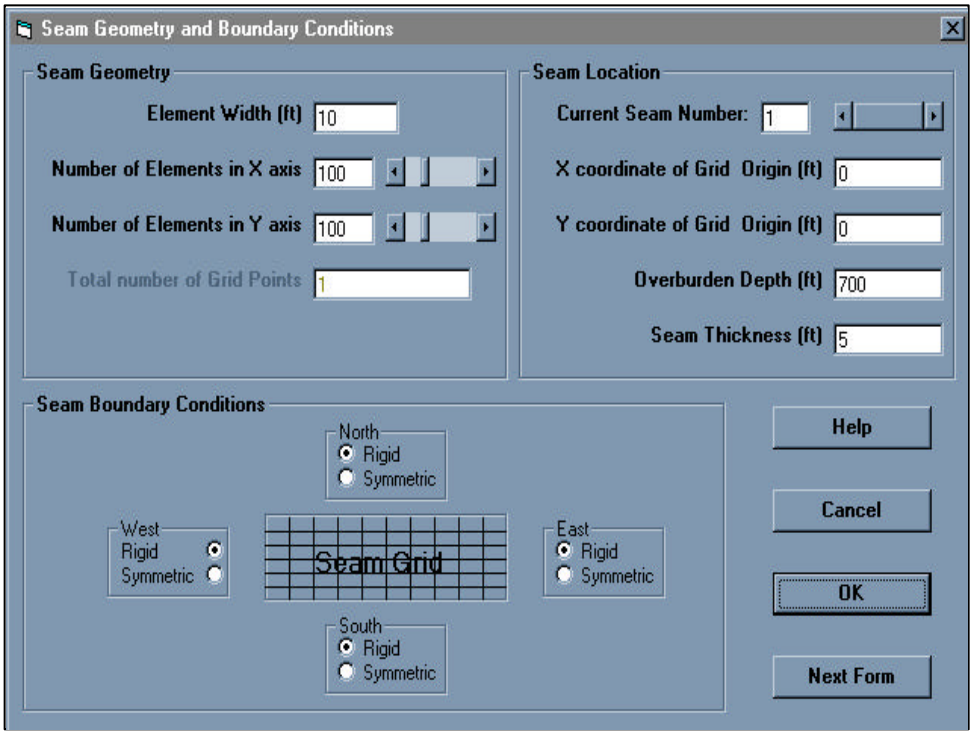
"100" elements in the X direction and

"100" elements in the Y direction.

We only have **"1"** seam, and we have no particular mine coordinates to use so we will leave the

"0" for the X coordinate and

"0" for the Y coordinate



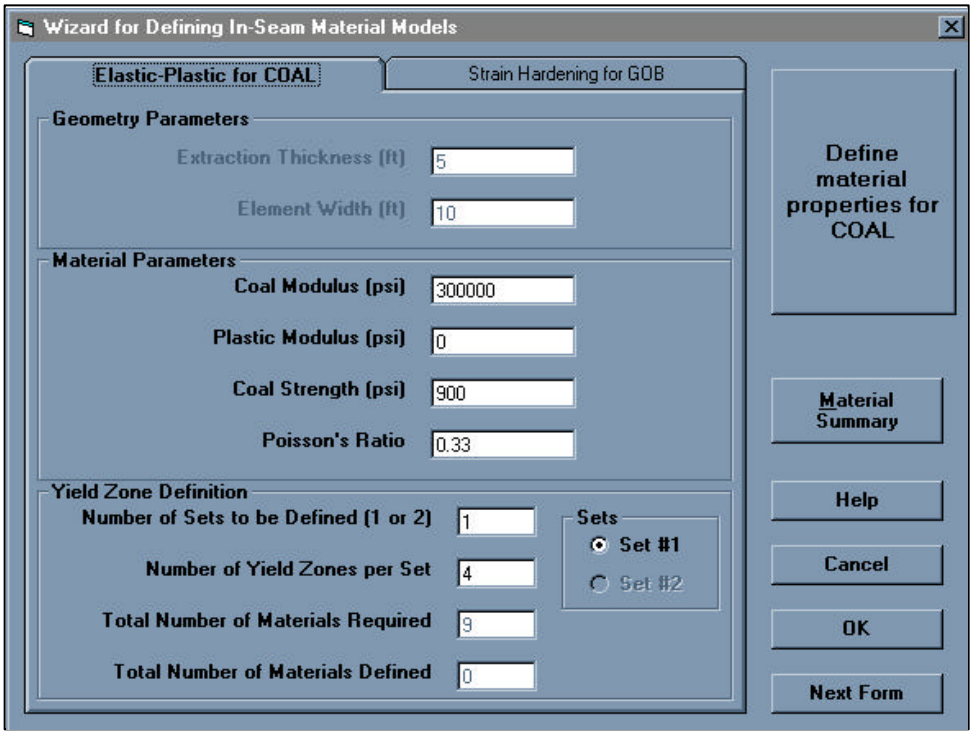
At this site we are

“700” ft deep with a

“5” ft seam thickness

Next, we need boundary conditions. These are the conditions at the edge of the model grid. It could be **“Rigid”** at the edge of the grid, in which case the material outside the grid will have zero convergence and will effectively support the roof around the grid edge, or we could specify **“Symmetric”,** in which case the material outside of the grid is a mirror image of what is inside the grid. For simplicity, we will go with the default **“Rigid”** conditions.

Click on: **Next Form** to contiune

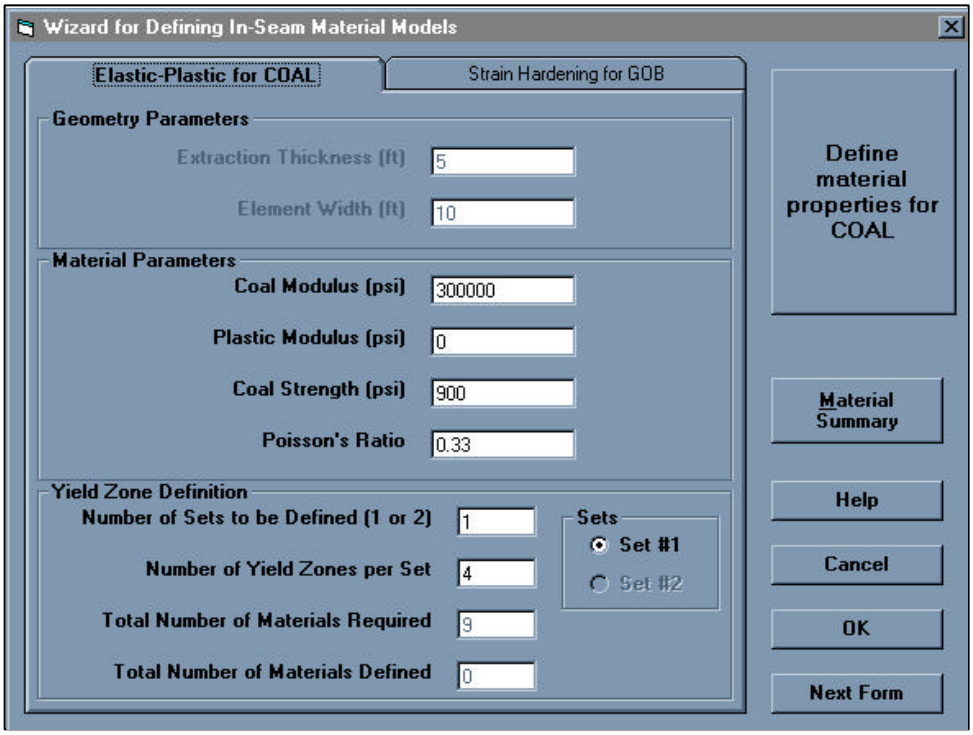


This opens the **"Wizard for Defining In-Seam Material Models"** window and starts on the **"Elastic-Plastic for COAL"** form.

This form is intended to help the novice user specify reasonable yielding coal properties.

You can see that we have default parameters for: **"Coal Modulus", "Plastic Modulus", "Coal Strength",** and **"Poisson's Ratio".**

Also, we can specify the **"Number of Yield Sets",** and the **"Number of Yield Zones per Set"**



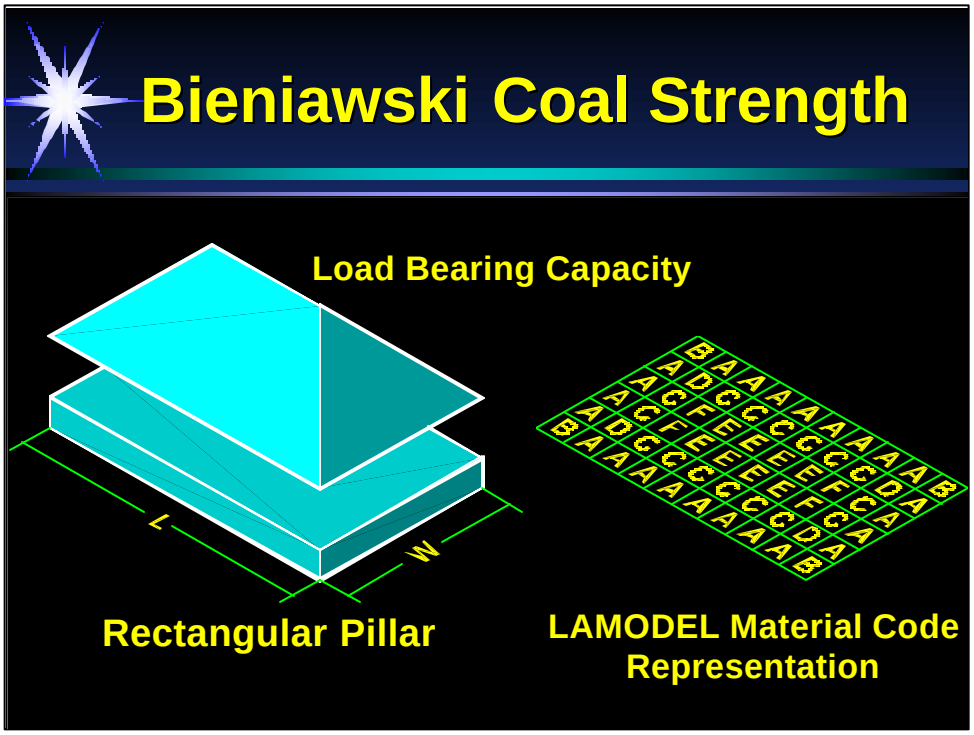
For the **"Material Parameters",** we will just use the default values:

"300000" for Coal Modulus

"0" for Plastic Modulus

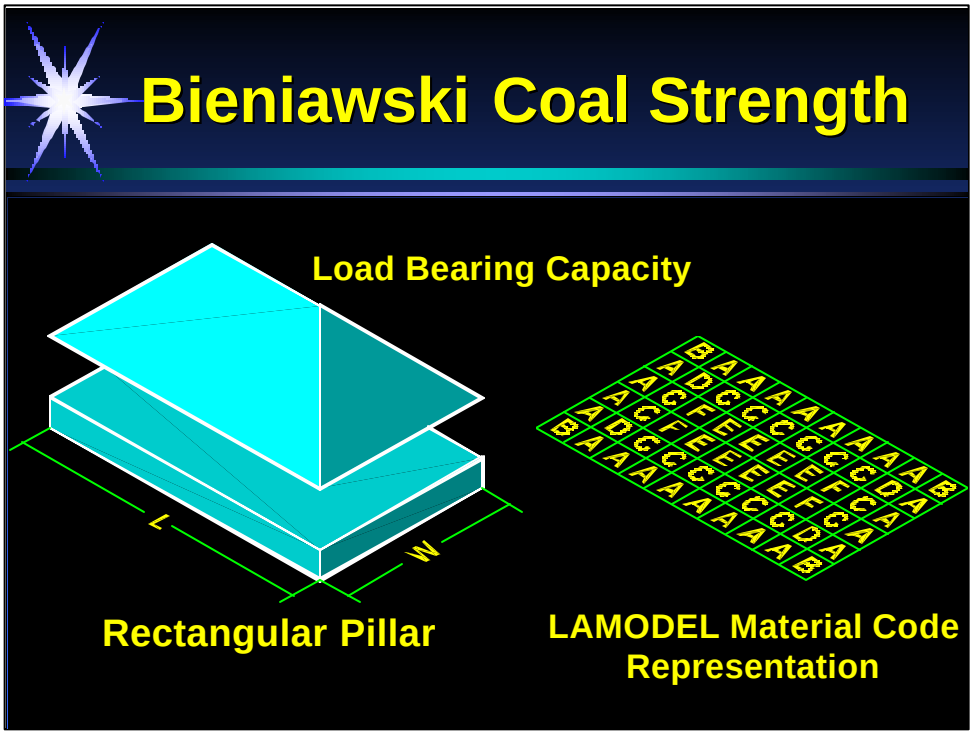
"900" for Coal Strength

"0.33" for Poisson's ratio



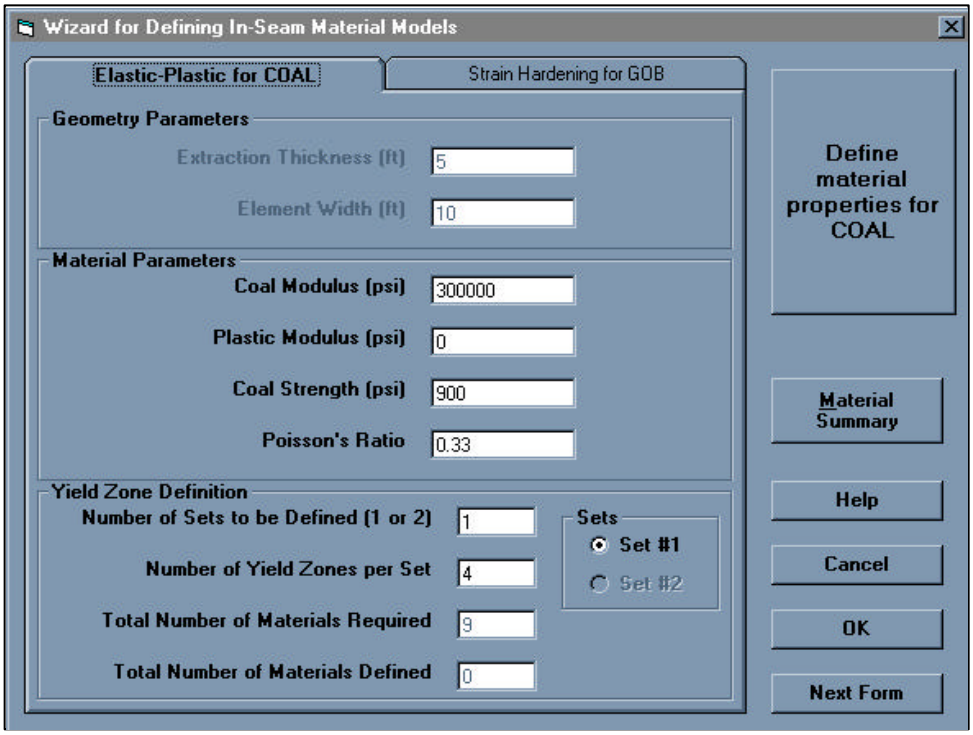
When the Bieniawski coal pillar strength formula is implemented in LAMODEL, the elements in a pillar increase in strength in proportion to the distance from the edge of the pillar. This strength increase is nominally due to the confinement provided by the surrounding coal.

As in the figure above, we end up with concentric zones of material around the pillars (two materials per zone, an edge and a corner material). The coal wizard helps us automatically define the coal properties needed to fill these concentric zones.



When we talk about Yield Sets, we are referring to sets of coal properties for different seams and/or coal thicknesses. In our case, we only have one seam, so we only need **"1"** yield set.

When we talk about Yield Zones, we are referring to the number of concentric rings of yielding material around a pillar that we want to use for the yield zone. We need to define a yield zone that will encompass the expected distance of yielding into the pillar. In our case, we will use a **"4"** element thickness (40 ft). We should not have any yield deeper than this into the pillar.



We are now ready for the program to calculate the necessary values of the coal properties.

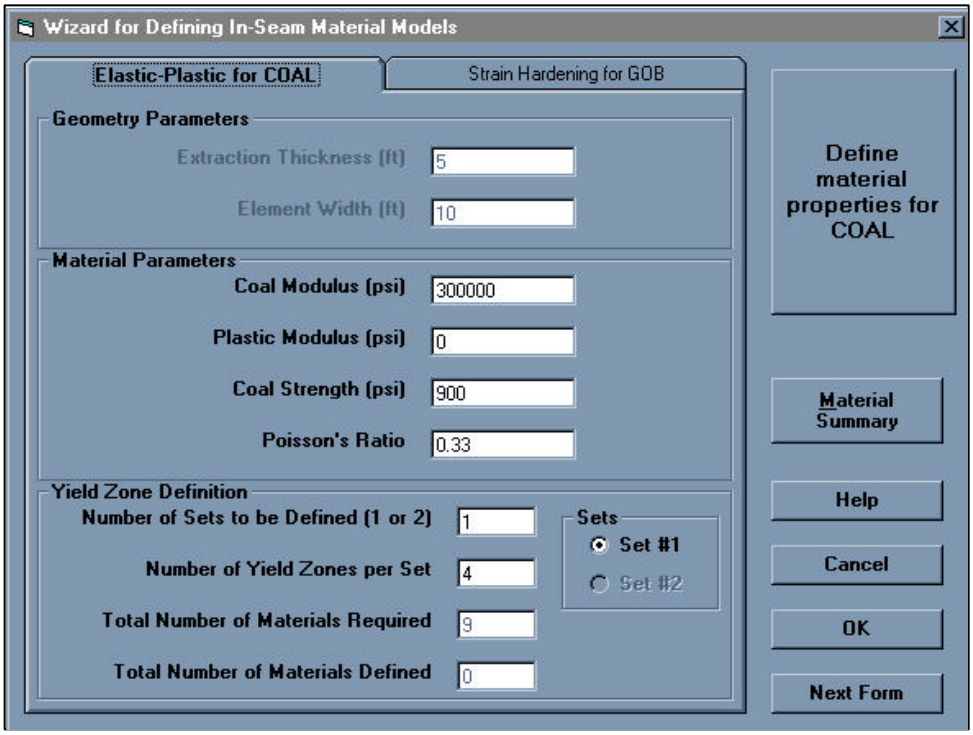
We click on: **“Define material properties for Coal”.**

This opens a message window that tells us what will happen and asks to continue.

We click **“Yes”**

Another message window appears telling us the calculation was successful and that the associated material codes are “A-I”.

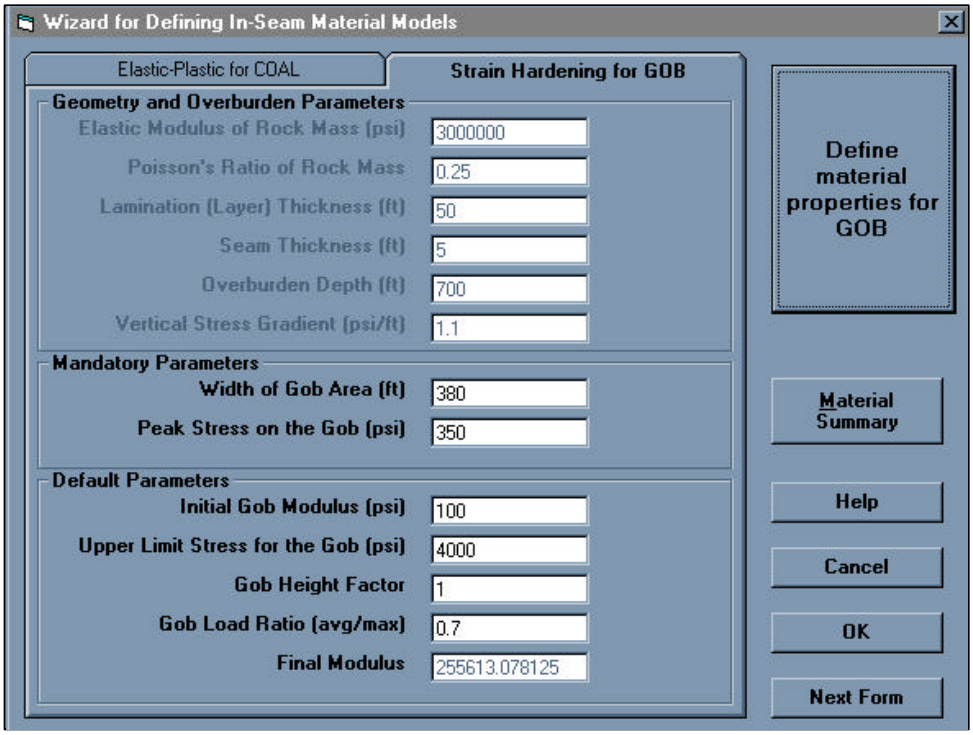
We click **“OK”** to continue



We have now created 9 materials to be used as yielding coal properties in the model grids. We will look at the parameters for these materials in a second, and when we edit the grids, we will see how these material are used.

For now, let's go generate gob properties by clicking on the tab:

"Strain Hardening for Gob"



This brings up the **“Strain Hardening for Gob”** form.

This form has a lot of parameters and looks fairly intimidating; however, we really only need to enter the two **“Mandatory Parameters“.**

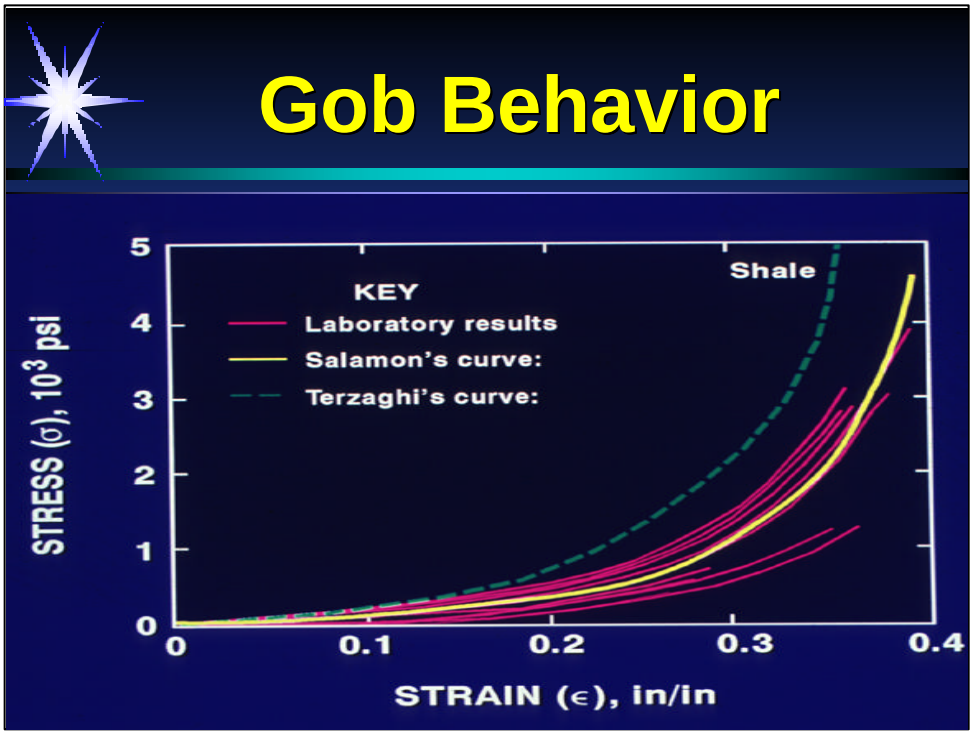
The first section, **“Geometry and Overburden Parameters”,** just lists parameters that were entered elsewhere in the program that are used in the calculation. The third section, **“Default Parameters”**, gives the suggested starting values for the gob material, and unless you are very experienced, these should not be adjusted.

Lab Test of Gob Material

The gob wizard is intended to simplify the task of determining gob material properties and to allow the user to obtain fairly accurate gob properties in the initial model run.

To accurately simulate the gob zone in LAMODEL, an exponential strain-hardening behavior is generally most appropriate and best matches that observed in the laboratory; therefore, this is the material model used by the Gob Wizard.

Gob Behavior

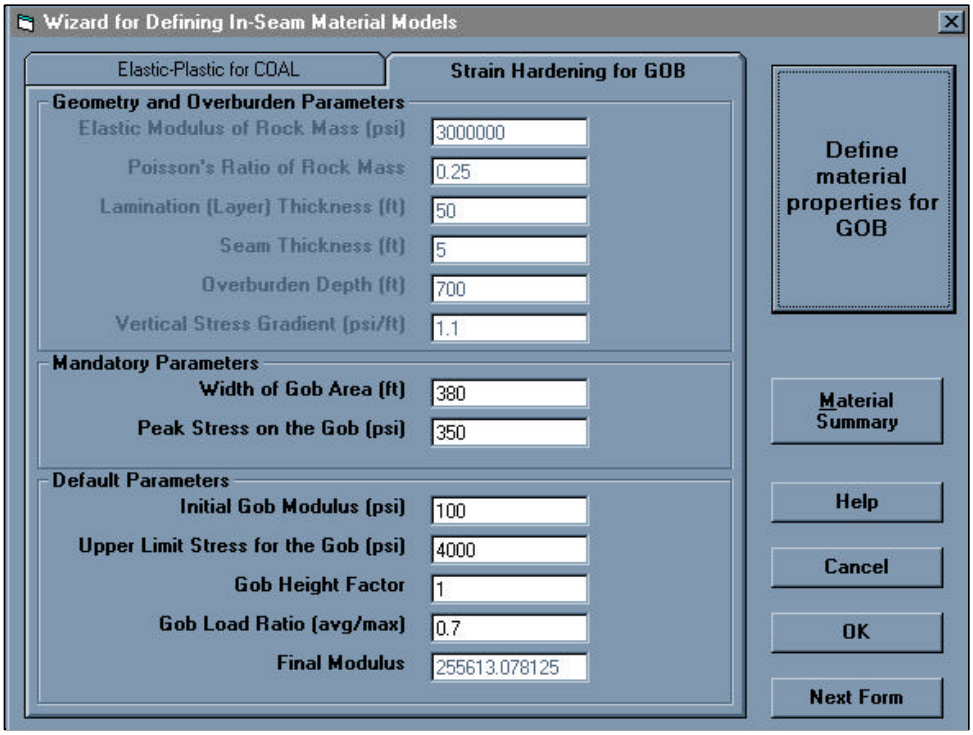


The Wizard's technique for pre-calculating accurate gob properties consists of determining a target level for the gob stress (based on overburden and gob width), then using the model parameters to approximately calculate expected convergence, and finally back-calculating the gob properties that provide the desired gob stress and convergence using a simplified two dimensional calculation.

The critical parameters that we need to enter are:

The **"Width of the Gob Area"** and the

"Peak Stress on the Gob"

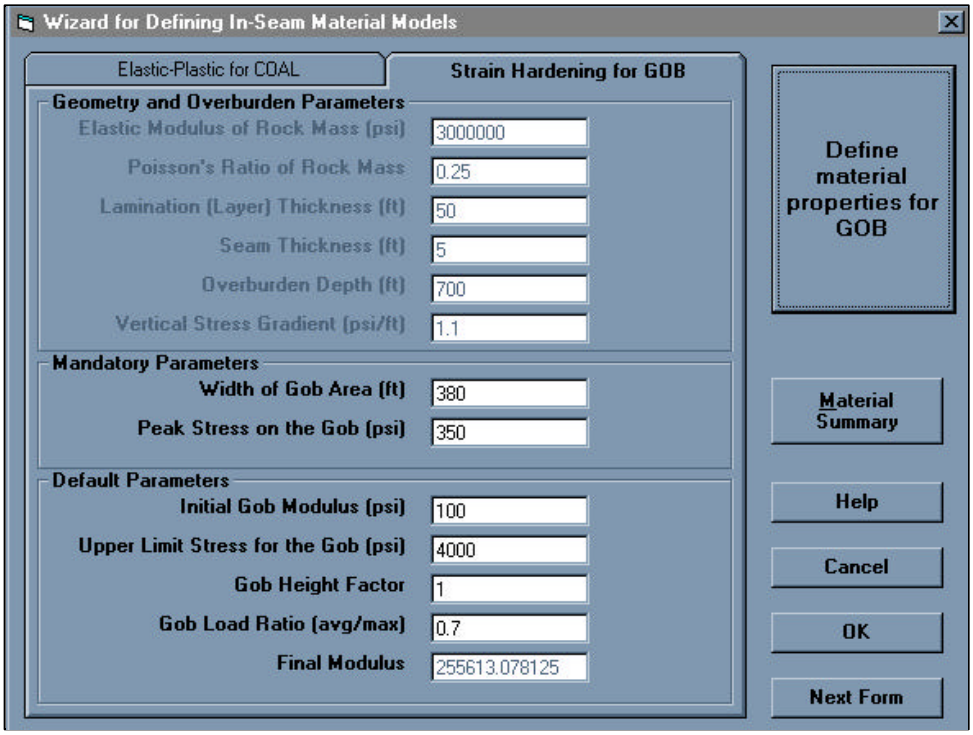


So, based on the geometry of our model, we will have a gob area that is **“360”** ft wide (6 x 60).

And using a little engineering judgment/experience we enter an expected peak stress of **“350”** psi for the 360 ft wide 700 ft deep gob area.

We are now ready for the program to calculate the necessary values of the gob properties.

We click on: **“Define material properties for Gob”.**



This opens a message window that tells us what will happen and asks to continue.

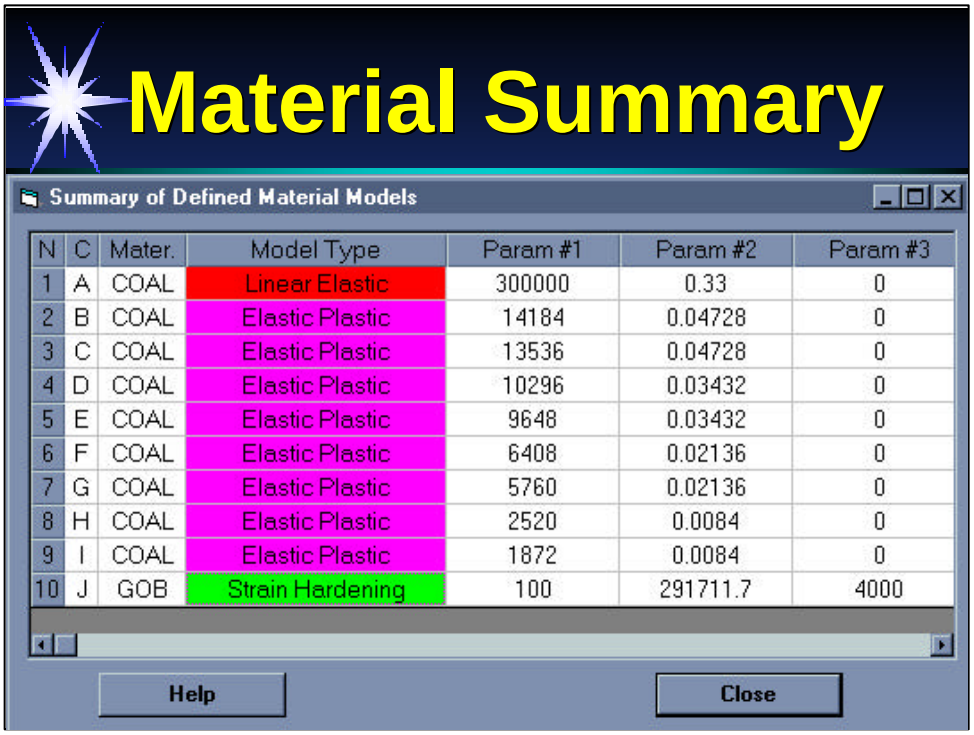
We click **“Yes”**

Another message window appears asking us if we want to append the gob material to our present set of materials.

We click **“YES”**

A final message window appears telling us that the gob material was defined.

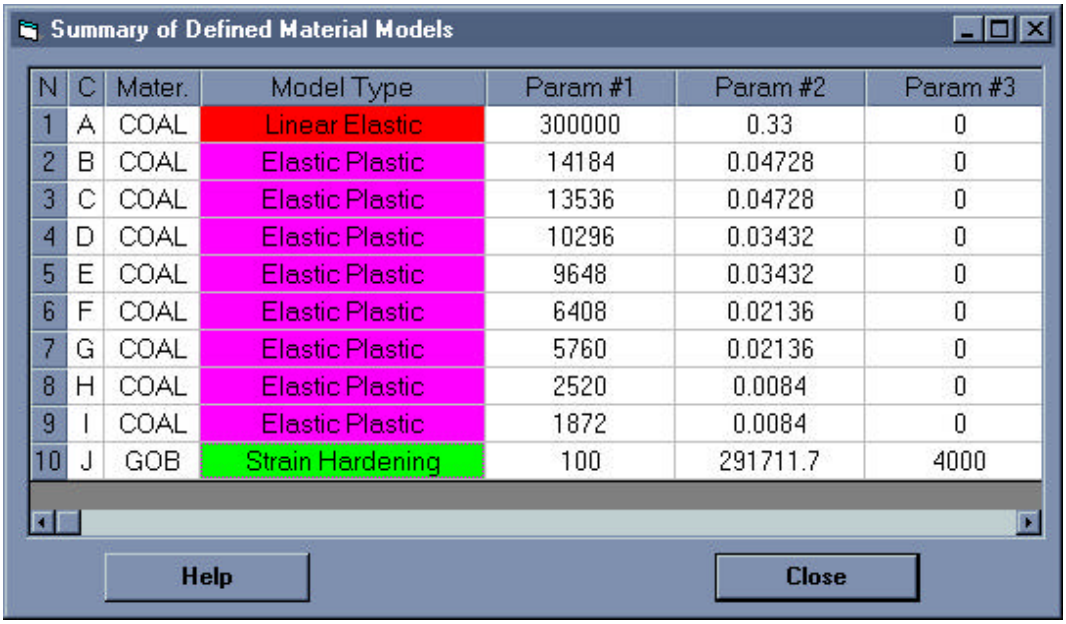
We click **“OK”** to continue



So let's take a look at the materials that we have defined.

Click on the **"Material Summary"** button and a window appears showing us the present materials and their parameters.

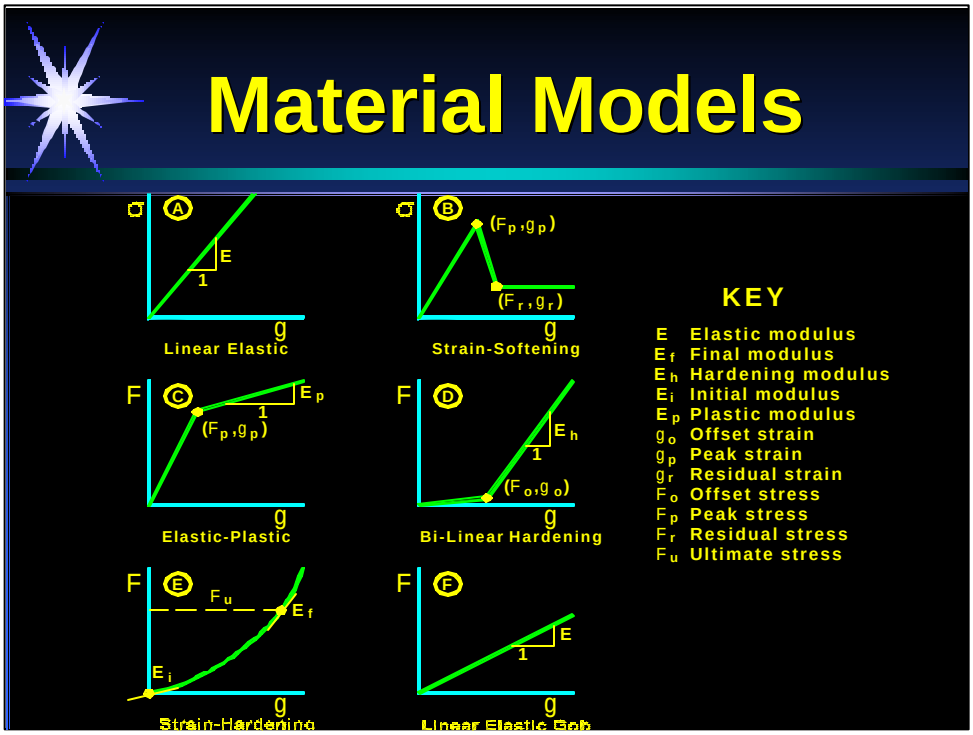
We see material "A" is Linear Elastic with parameter #1 = 300000 (the Elastic Modulus) and parameter #2 = 0.33 (the Poisson's Ratio). This is the core material for our yielding coal properties.



We also see the Elastic Plastic materials for the yield zones.

For the Elastic Plastic material, Parameter #1 is the Peak Stress, Parameter #2 is the Peak Strain and Parameter #3 is the Plastic Modulus.

At the bottom of the page, we see that material “J” is the gob material and we see the parameter values for it.

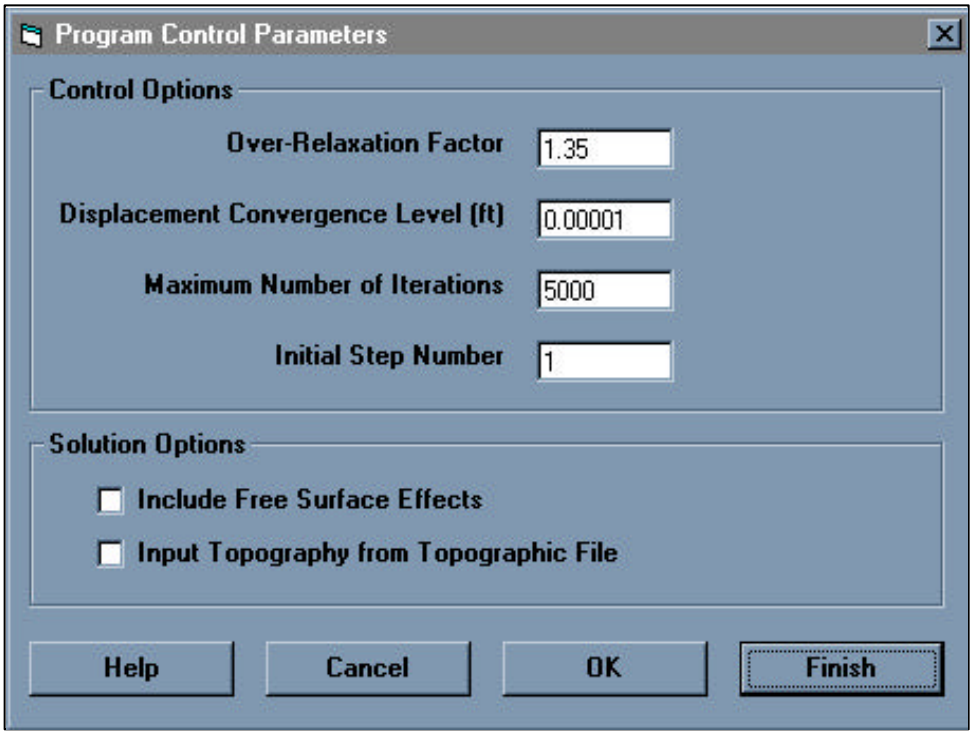


If we want to know more about the material models and the parameters that are necessary for defining the materials, a complete description of each of the materials is provided in the help file.

This figure provides a quick reference.

To get out of the **“Material Summary”,** we click the **Close** button

and to continue entering parameters, we click on **Next Form**

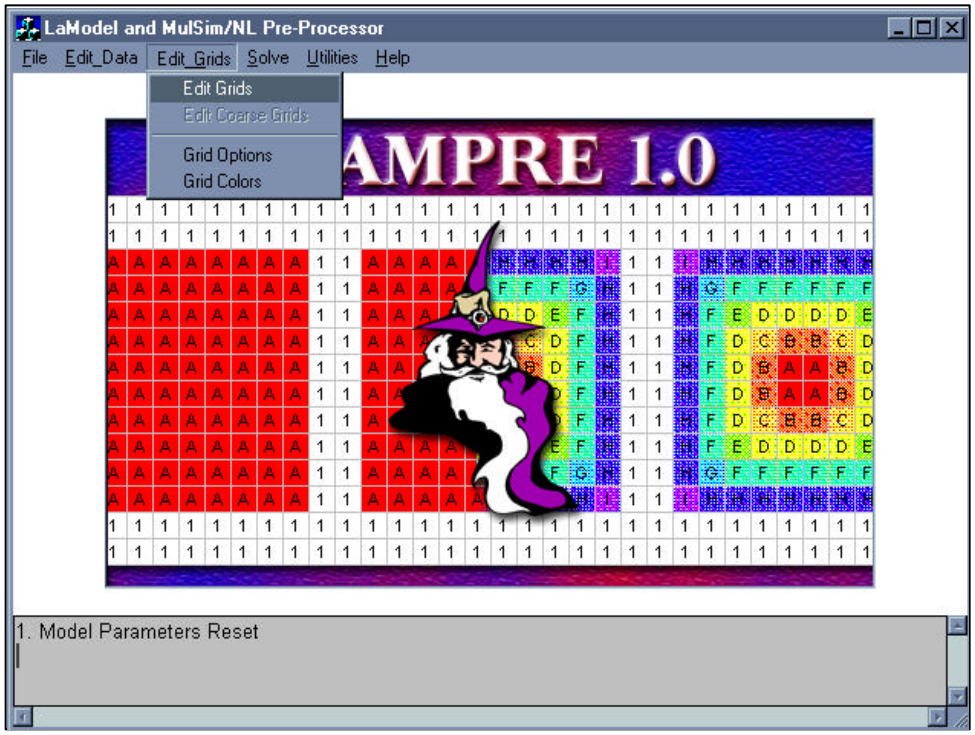


This opens the **"Program Control Parameters"** form. This is the last form for entering parameters.

The parameters in this form control the calculation of the stresses and displacements in the LAMODEL program.

In general, the default values will work for the majority of models and we will use these for our example.

To finish entering parameters, we click **Finish**

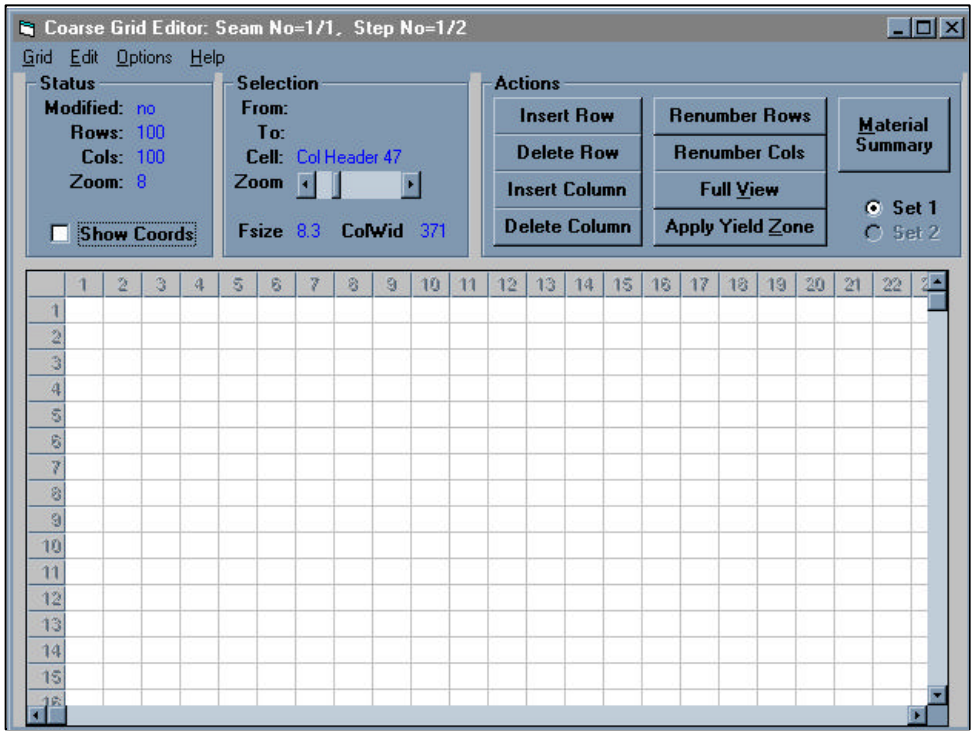


This brings us back to the main LAMPRE window.

Next we want to graphically enter the material codes for the seam grid. To do this, we click on:

Edit_Grids=>Edit_Grids from the menu bar,

The opens the **“Grid Editor”** window

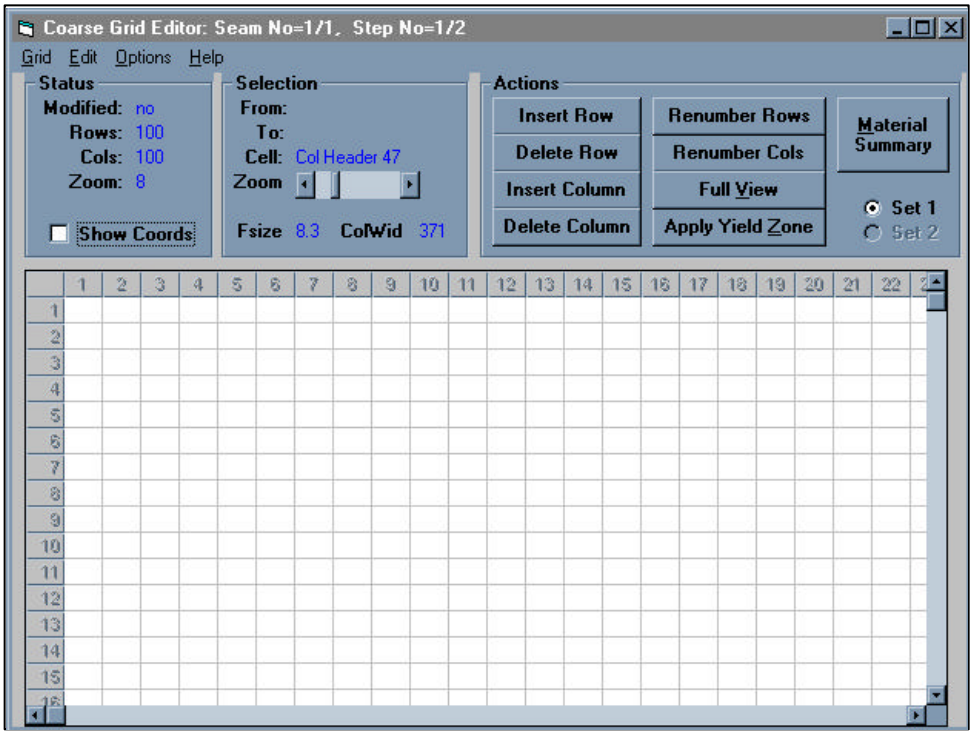


The Grid Editor essentially works like a spreadsheet.

Material codes for individual cells may be set by clicking on a cell and then typing and entering an appropriate letter from A to Z.

Blocks of cells may be changed by clicking and dragging the mouse over a range, and then typing and entering the material code.

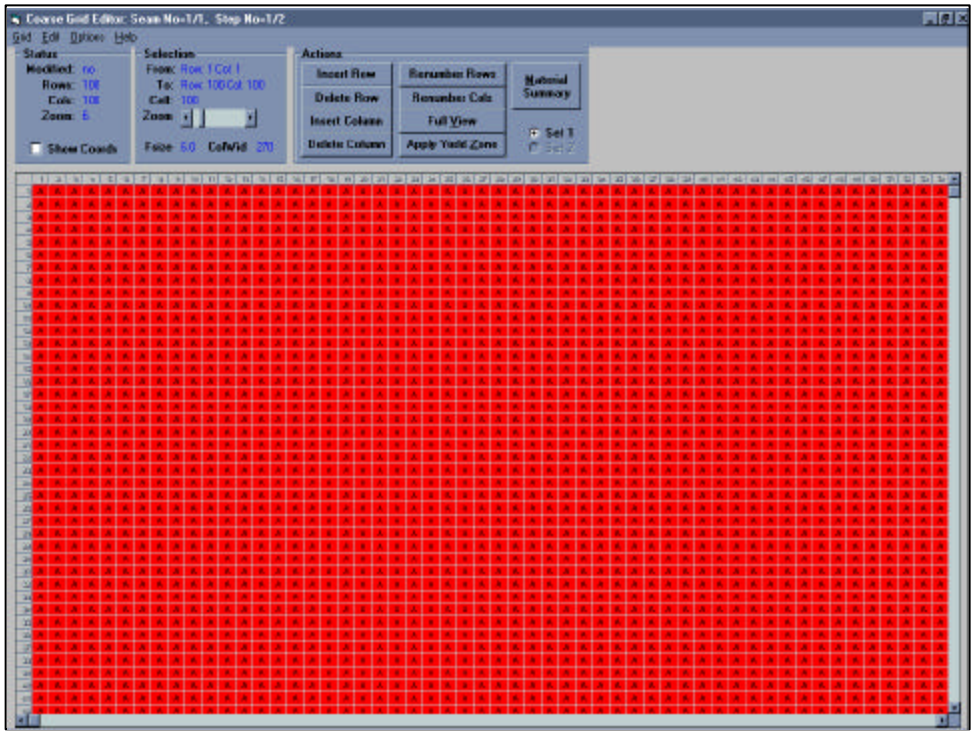
Or, entire rows/columns may be selected and changed using the row/column number.



Also, a block of cells may be copied and pasted (greatly simplifying the creation of numerous pillars).

Finally, the grid editor allows for automatic yield zone generation based on materials that were automatically defined using the coal property wizard

We will use all of these techniques to define the materials codes for our room-and-pillar production panel.

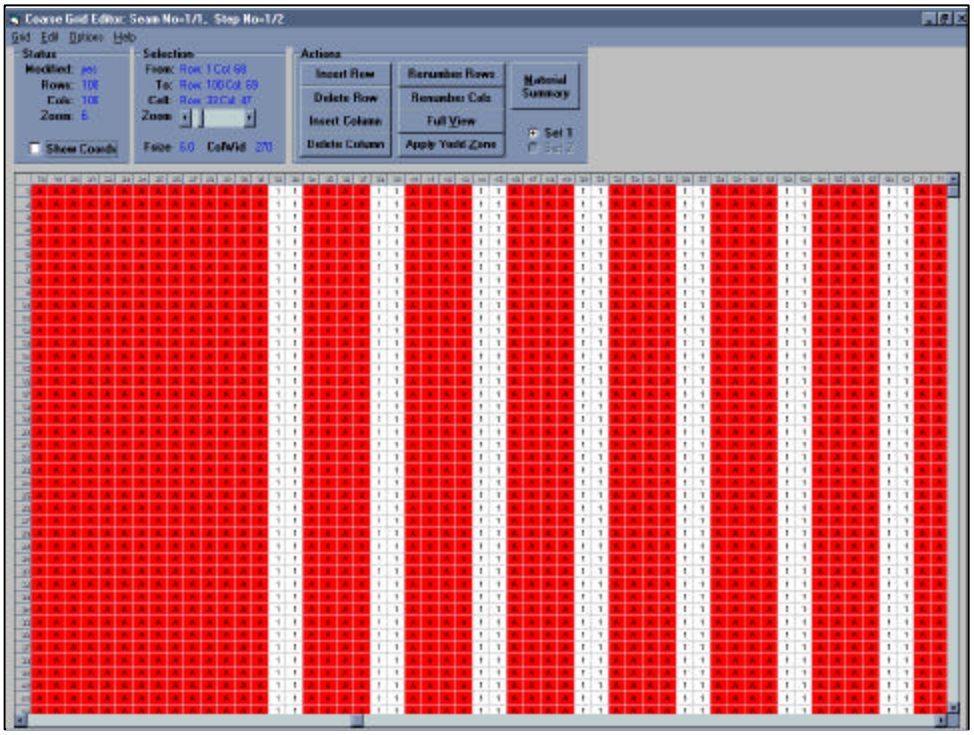


First, let’s zoom out a bit by clicking twice on the **left arrow** on the zoom slider.

Then, Let’s make the window full screen by clicking on the **middle button** in the **upper right corner** of the window.

Now, let’s fill the entire grid with the elastic coal material **“A”**

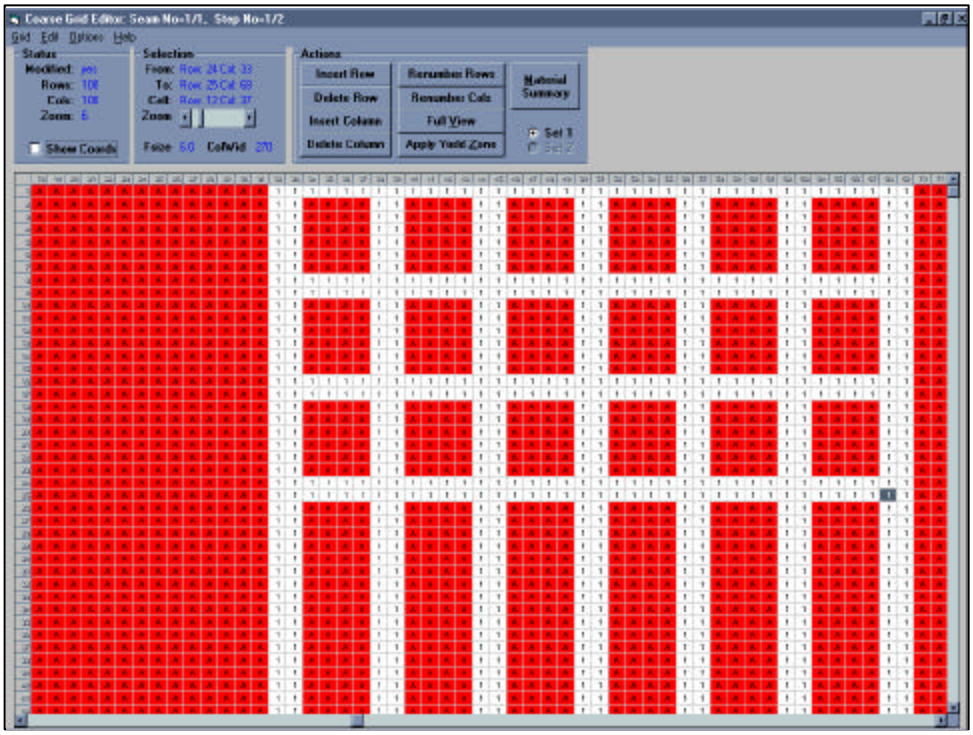
To do this, click in cell **(1,1)** and drag to cell **(100,100)**, (if you drag off of the screen, the window will scroll). Now enter an **“A”** from the keyboard and hit the **Enter** key.



Now, Let’s start defining the entries by selecting the headings for columns **32** and **33**.

Now enter a **“1”** from the keyboard and hit the **Enter** key.

Repeat this for columns **38-39, 44-45, 50-51, 56-57, 62-63, 68-69** to define 7 entries

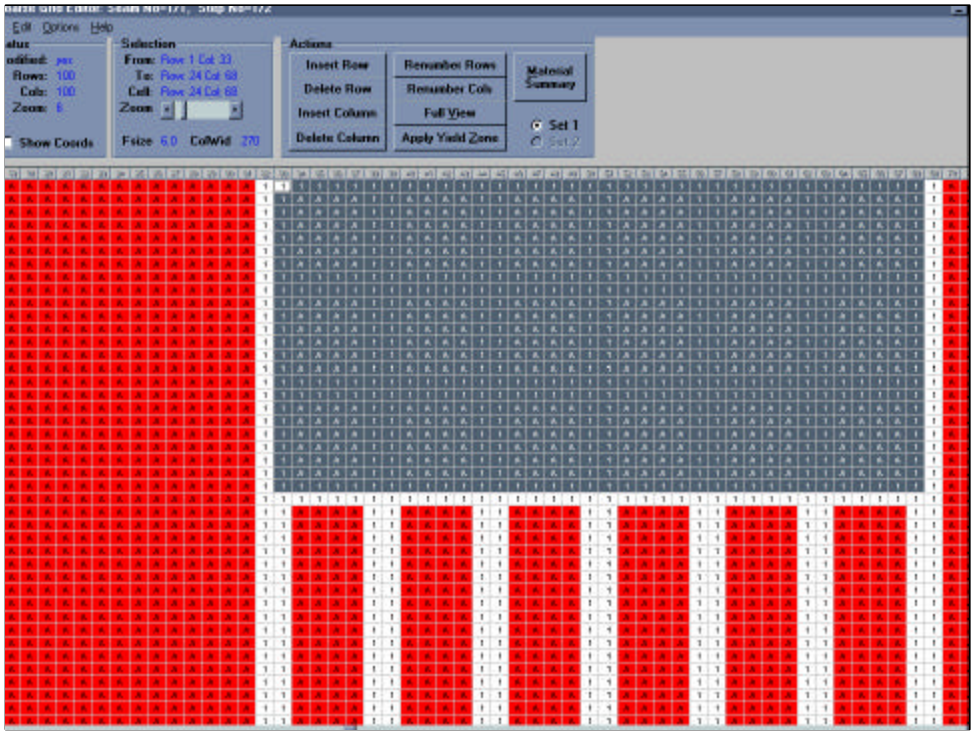


Next, Let's start defining a couple of crosscuts.

To do this, click in cell **(1,33)** and drag to cell **(1,68)**. Now enter a **"1"** from the keyboard and hit the **Enter** key. This creates half a crosscut at the top of the screen.

Now, click in cell **(8,33)** and drag to cell **(9,68)**. Enter a **"1"** from the keyboard and hit the **Enter** key. This creates a full crosscut with 40 by 60 ft pillars

Repeat for a couple more crosscuts.



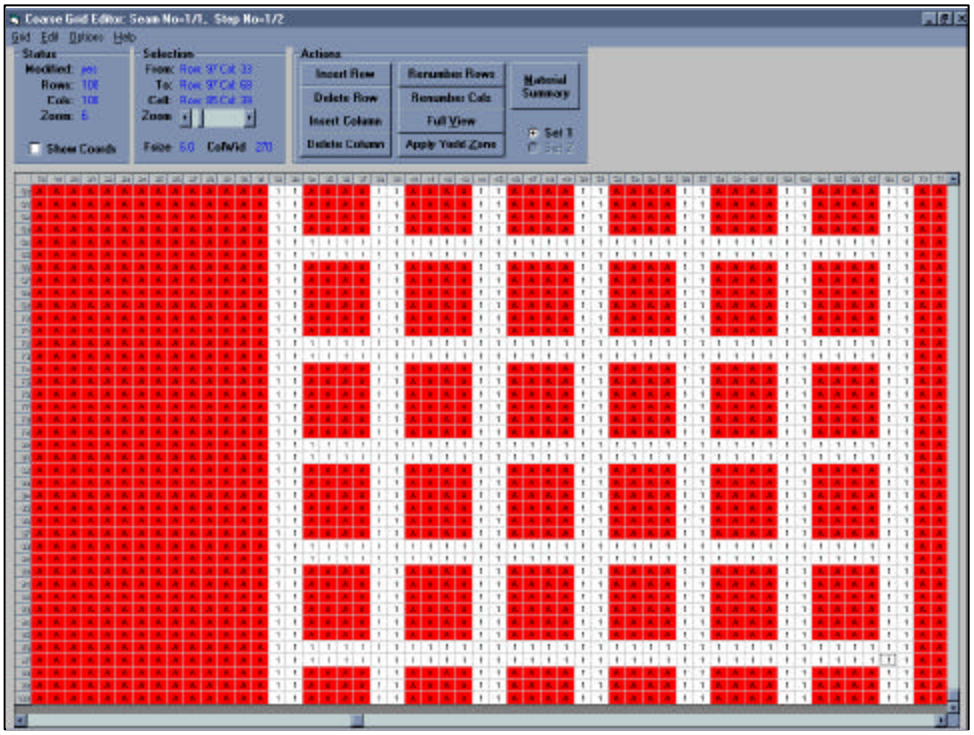
Next, Let's copy these three rows of pillars the rest of the way down the grid

To do this, click in cell **(1,33)** and drag to cell **(24,68)**.

Now go to the **Edit** menu and select **Copy Range**.

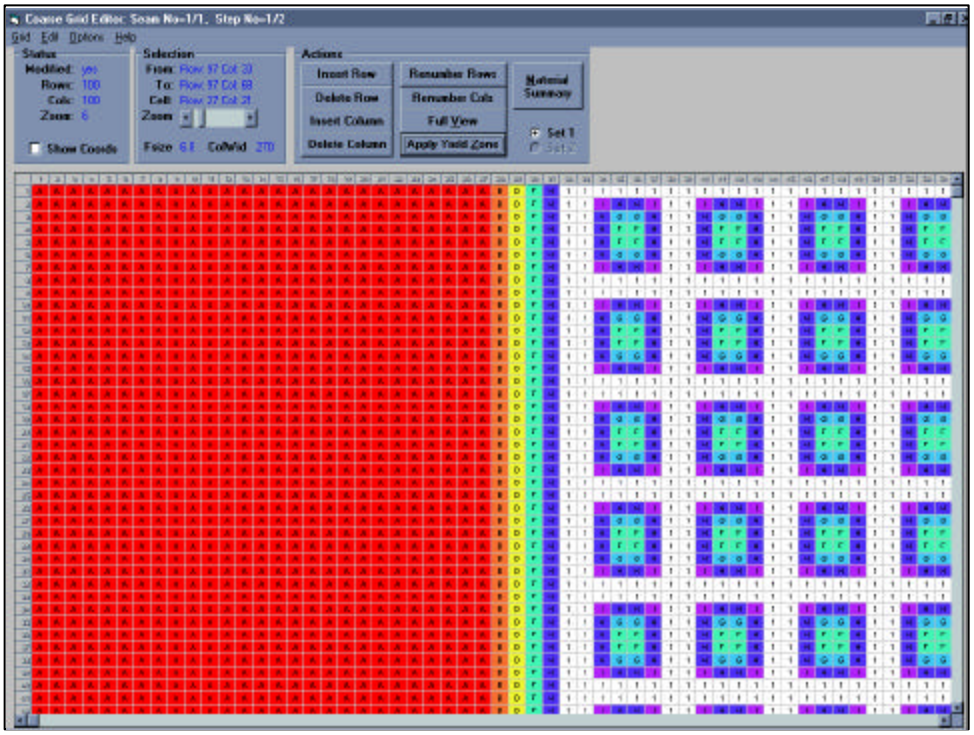
Next, click in cell **(25,33)** and drag **at least** to cell **(48,68)**. (You can cover more than the copy area and only the copied area will be inserted from the top left corner)

Now click **Edit=>Paste Range**



Continue copying and pasting the pillars the rest of the way down the grid.

You should end up with exactly half a pillar at the bottom of the grid.

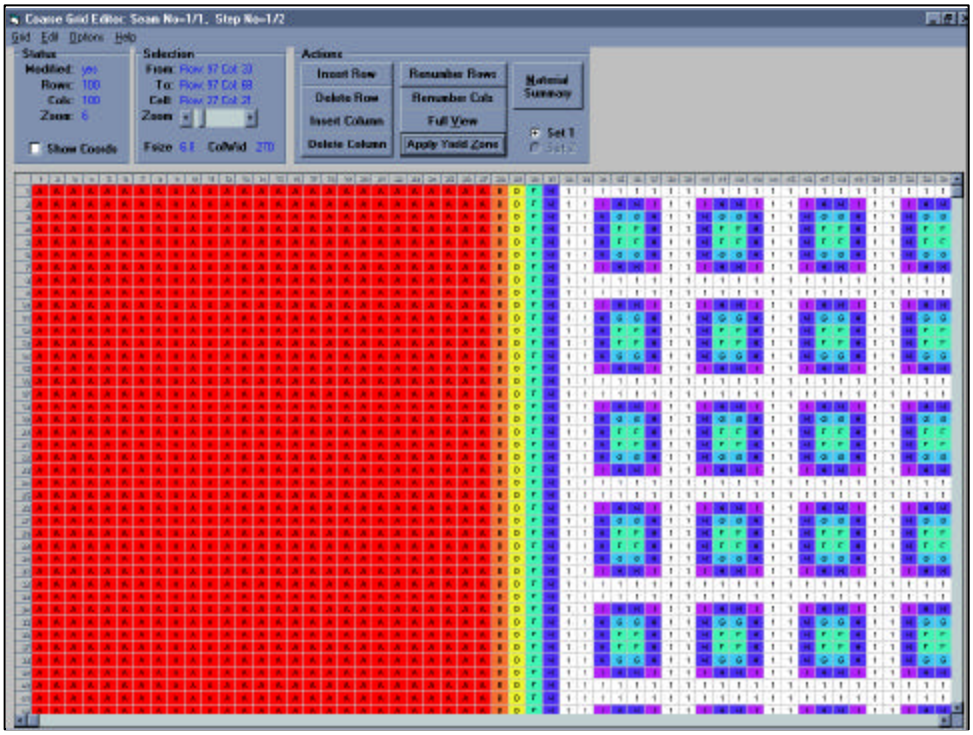


Next, scroll to the upper right corner of the grid, and let's apply the yield zones.

Click on the **"Apply Yield Zone"** button

A message appears describing the yield zone to be applied. Click **"Yes"** to continue.

The material properties around the coal edges are change to reflect the defined yield zone

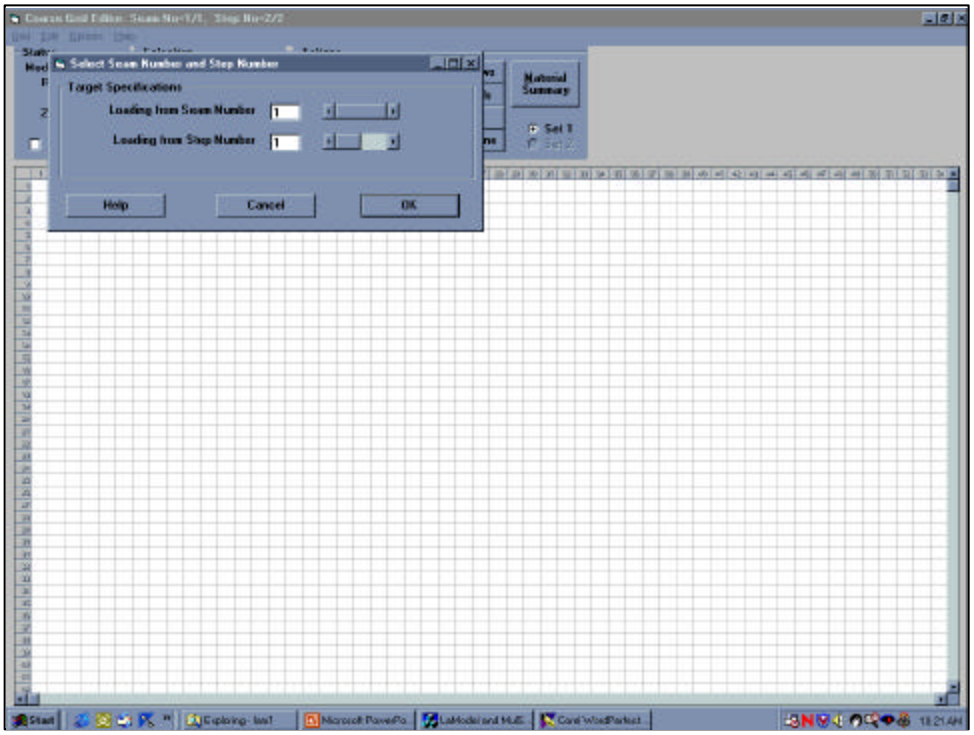


Now, let’s work on the second step.

Click **Grid=>Save** to save the grid for the first step.

Click **Grid=>Next Step** to bring up the empty second step grid.

Click **Grid=>Load From** to load the first step grid into the second step grid.

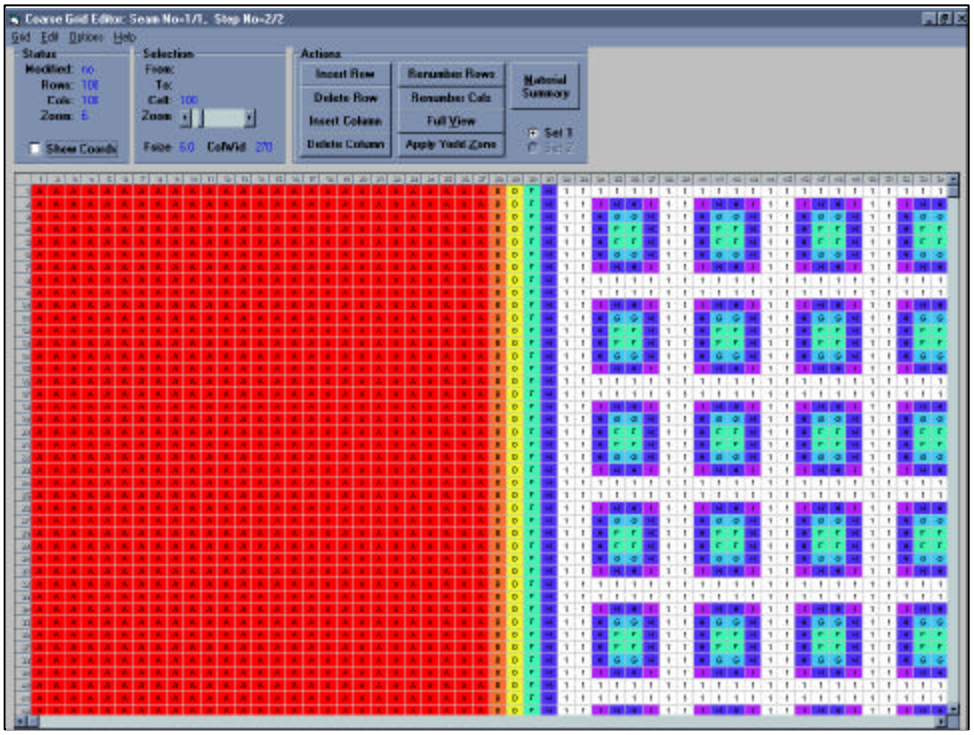


Now, let's work on the second step.

Click **Grid=>Save** to save the grid for the first step.

Click **Grid=>Next Step** to bring up the empty second step grid.

Click **Grid=>Load From** to load the first step grid into the second step grid.



This opens a form for selecting the seam and step number for the grid to load

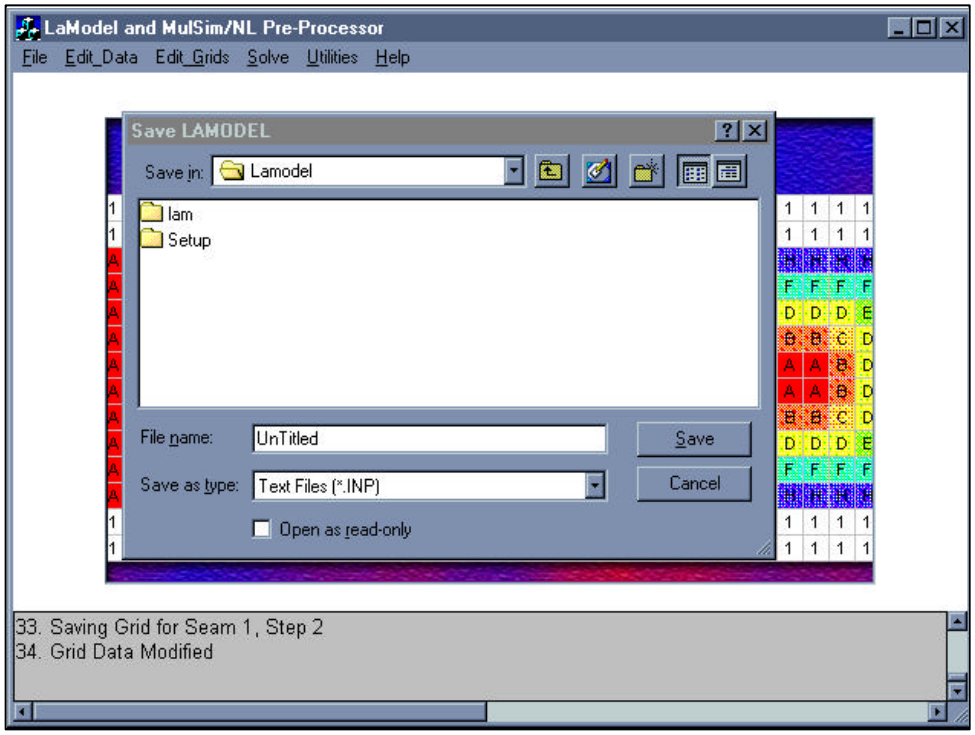
Select:

“1” for the Seam Number, and

“1” for the Step Number

Click **“OK”** to continue

This fills step 2 with the grid from step 1.



Let's go ahead and save this grid

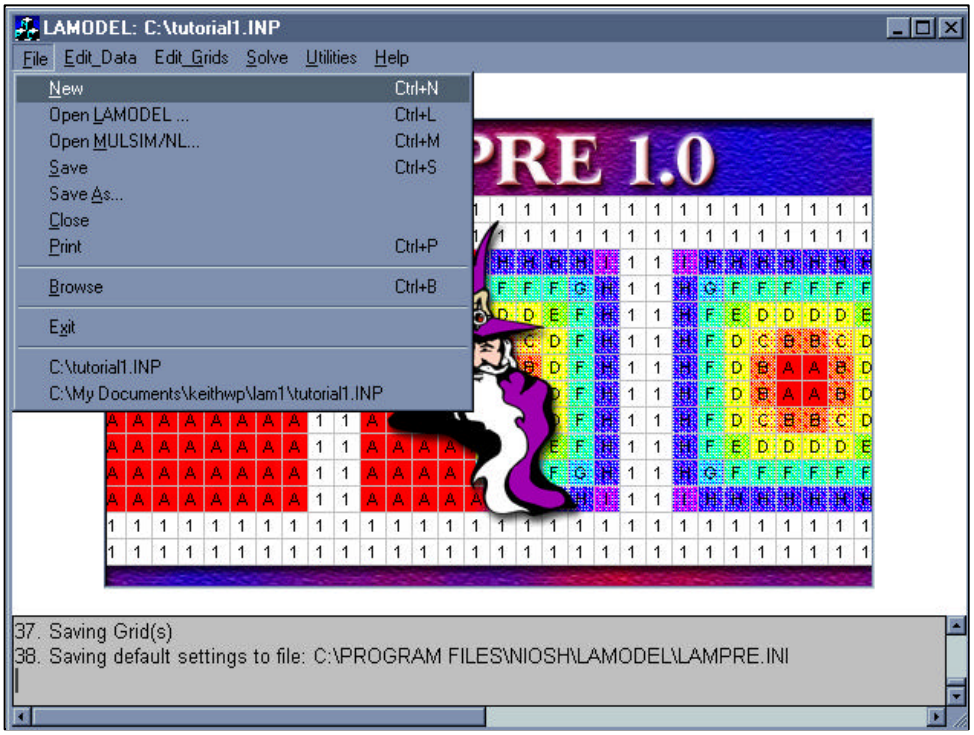
Click **Grid=>Save,** then exit from the grid editor.

Click **Grid=>Exit**

This brings up the main LAMPRE window. Let's save this input file as "Tutorial1.inp"

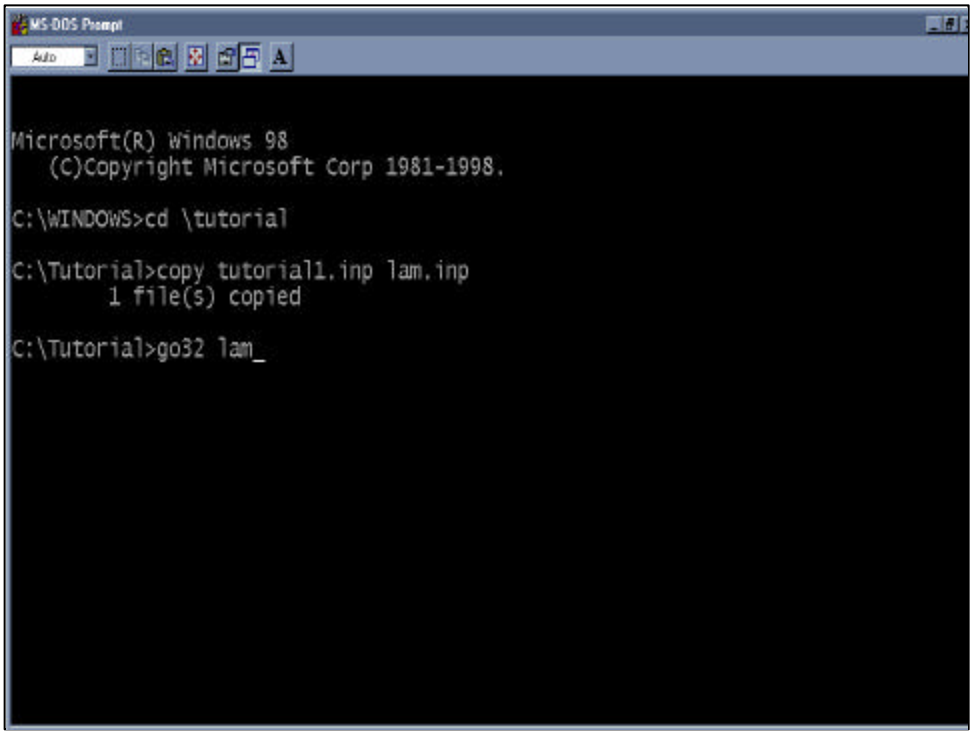
Click **File=>Save As**…,

In the Save LAMODEL form, change to the desired directory and enter **"Tutorial1"** for the Filename



We are now ready to exit the preprocessor and run the program

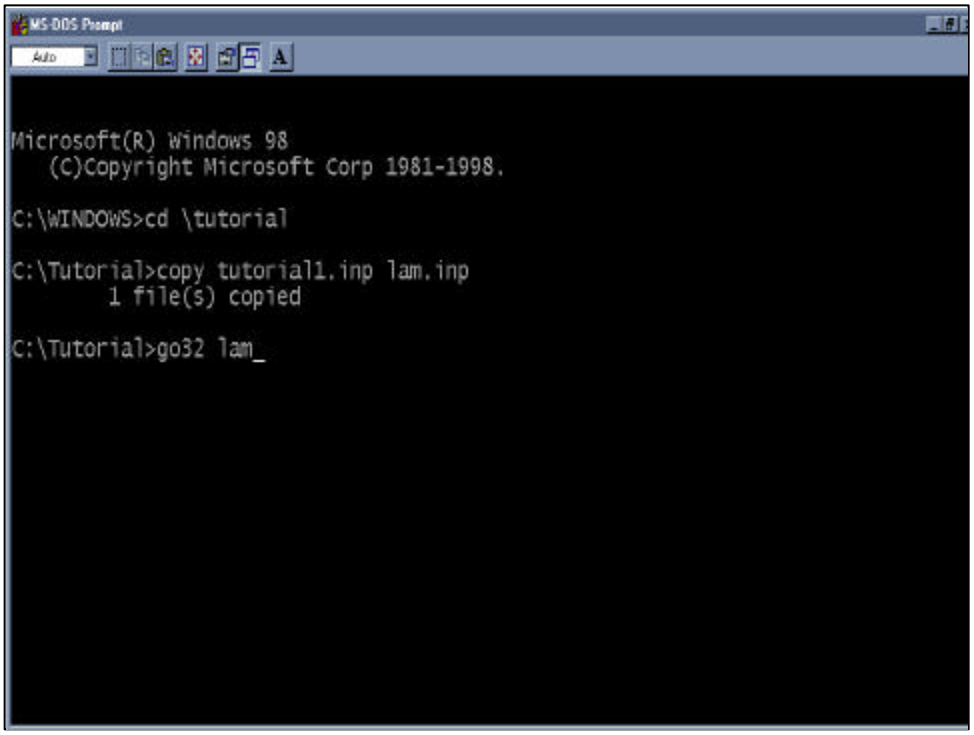
Click **File=>Exit,**



Once the input file is created, the LAMODEL numerical analysis is run in the a batch mode in DOS for calculating the stress and displacements at the seam level.

Model runs can take several minutes to several days depending on the computer speed and model complexity (which includes such factors as; number of steps, number of seams, and grid size).

The output from the calculation phase is stored in a data file for subsequent analysis by the post-processing program, LAMPLT.



To run the input file that was just created, bring up a DOS window (only compatible with Windows 95/98, **not NT**).

Start=>Programs=>DOS Prompt

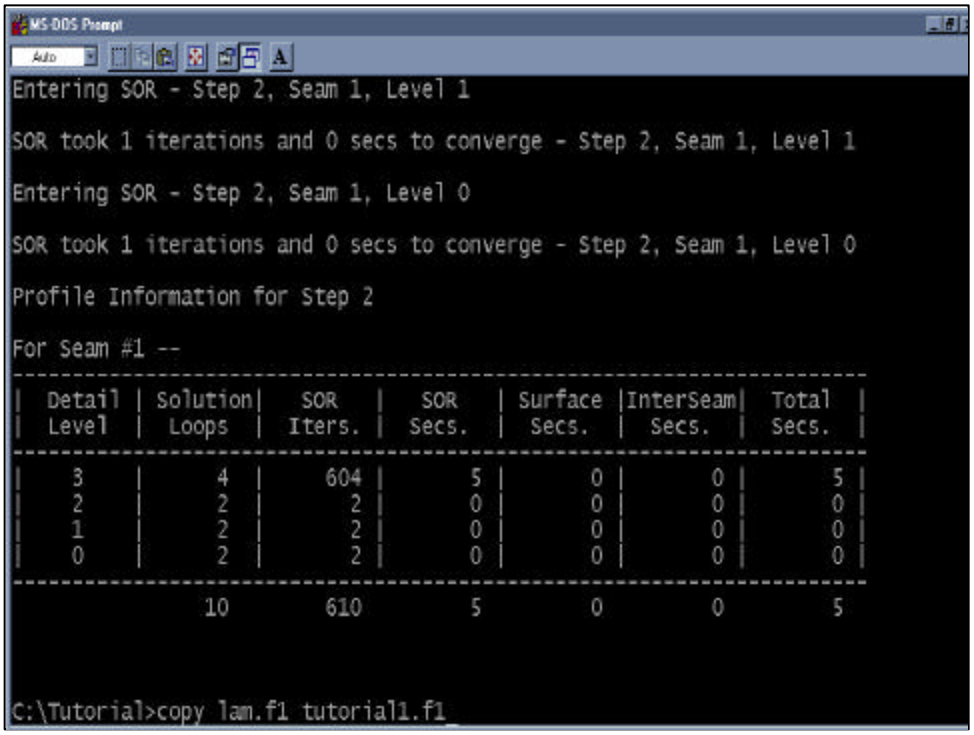
Change to the directory with the input file and the LAMODEL program:

cd \tutorial

Copy the input file to the default input name:

Copy tutorial1.inp lam.inp

Enter **go32 lam** to run the program



While the program is running, information concerning the status of the convergence, the seam and step number, and the program running times is scrolled to the screen.

This general information is also stored in a file named:

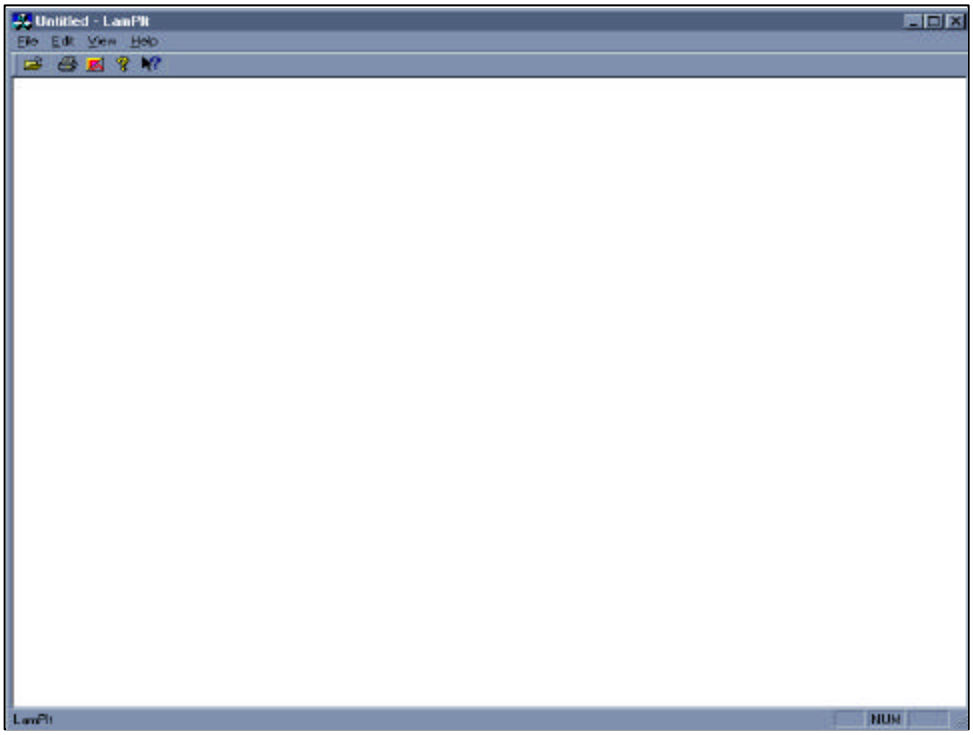
“lam.out”

The stress and displacements calculated by the program are stored in a file called:

“lam.f1”

To save the output, we will copy “lam.f1” to “tutorial1.f1”

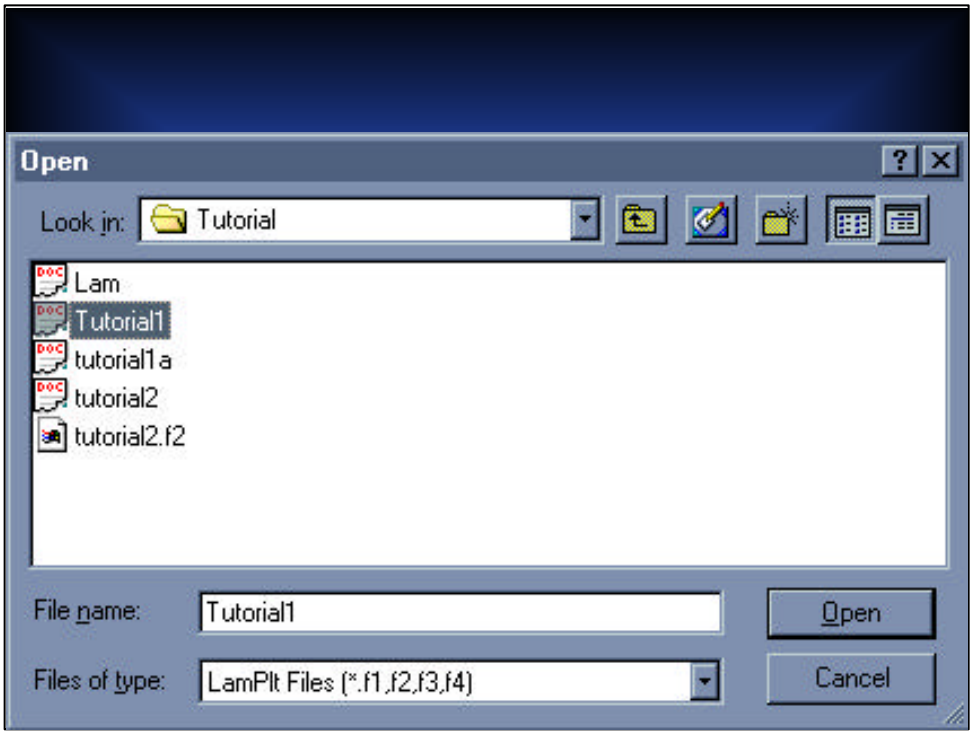
copy lam.f1 tutorial1.f1



To look at the output from LAMODEL, we need to start the postprocessor, LAMPLT.

Click on:
Start=>Programs=>NIOSH=>LAMODEL=>LAMPLT

This will start LAMPLT, and the main LAMPLT window will appear.



We now need to open the output file form LAMODEL

Click on:

File=>Open

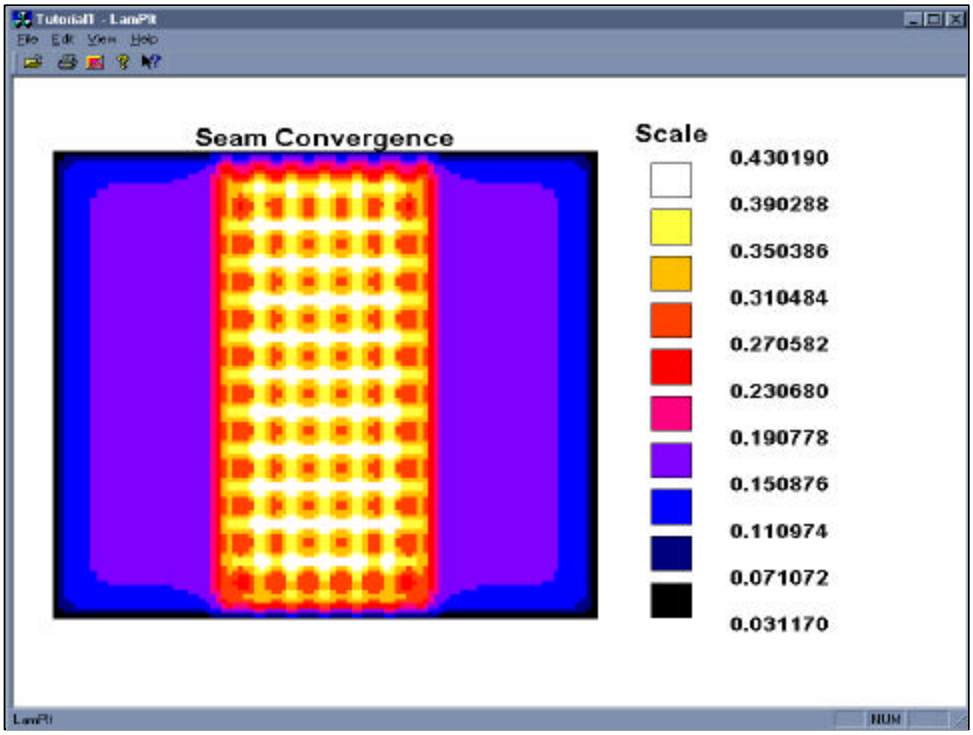
In the “Open” form:

change to the necessary directory

select the **“Tutorial1”** file,

Click **“Open”**

This will open the Tutorial file and bring up a 3D plot of the convergence.



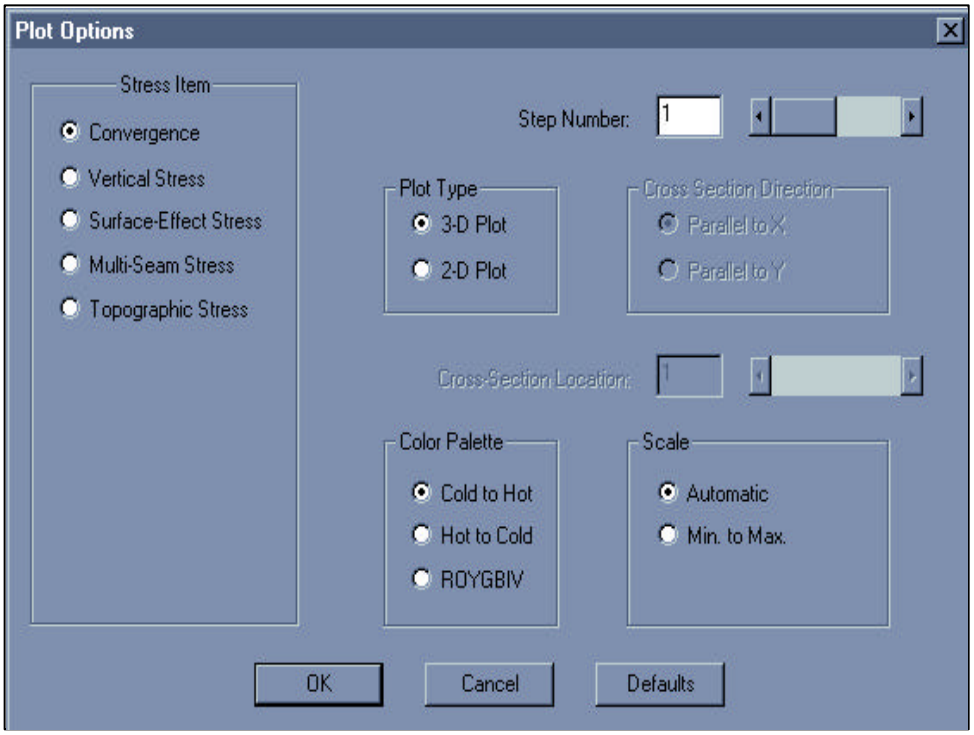
This is a “typical” LAMPLT colored square, or pseudo 3D, plot.

For this plot, the desired value for each element in the grid is plotted as the appropriate scaled color.

The majority of the analysis in LAMPLT is accomplished through the **“Plot Options”** form

To get to this form, click:

Edit=>Plot Options…



Using this form, we can:

Examine various **Stress Items,**

Change the **Plot Type,**

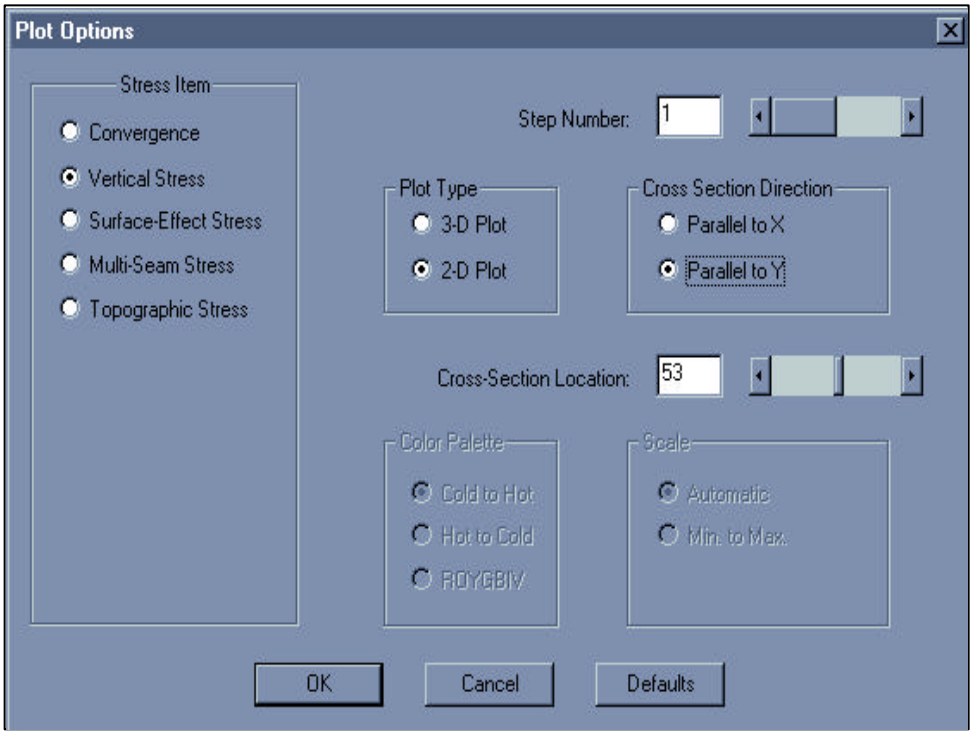
Set the **Cross-Section Direction**,

Set the **Cross-Section Location,**

Adjust the **Color Palette**, or

Adjust the **Scale.**

Let’s look at a 2D cross-section of the vertical stress running up the pillars in the panel



We need to click on:

“Vertical Stress”

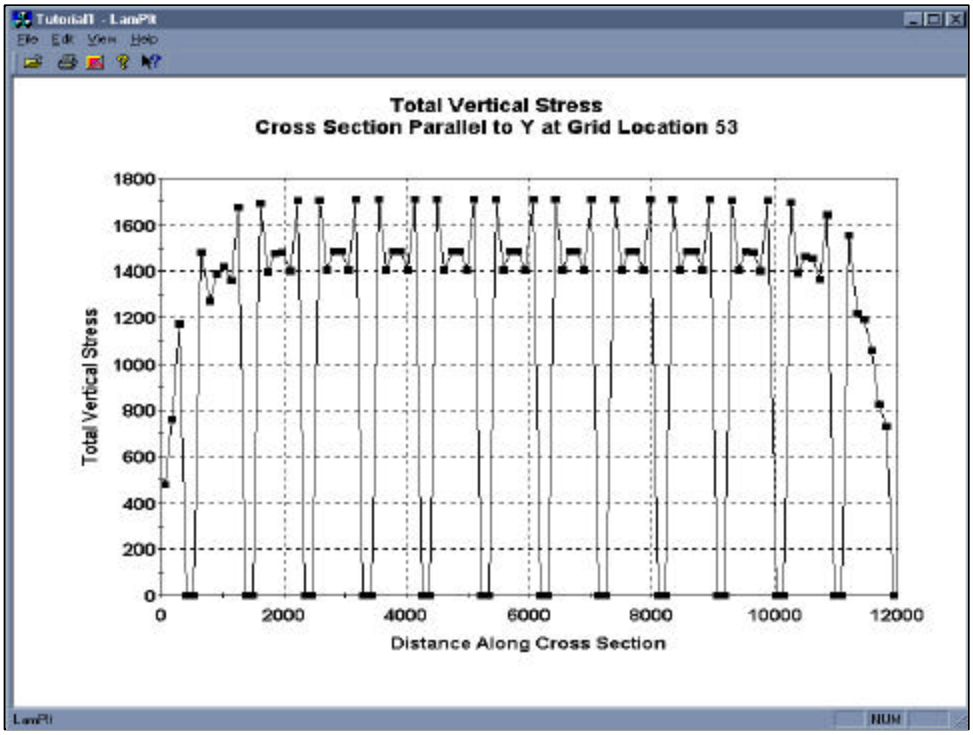
“2-D Plot”

“Parallel to Y”

“53” for the Cross-Section Location (a line up the middle of the fourth pillar row)

Then click:

“OK” to continue and make the new plot



In the main LAMPLT window, we now have a 2-D plot of the vertical stress at column 53.

(To edit these 2-D plots, you can right click in the window to bring up an extensive dialog box of editable properties.)

(Also, you can copy the graphics to the windows clipboard using **Edit=>Copy,** and then paste the graphics into most windows programs.)

This concludes the official tutorial, please try a number of other plot options to examine the second step, at you leisure.